CONTENTS

Page

1 MAZATROL PROGRAM CONFIGURATION 1-1

1-1 Program Configuration 1-1

2 PROGRAM COORDINATE SYSTEM 2-1

3 PROGRAM CREATION 3-1

3-1 Common Unit 3-5

3-1-1 Setting unit data (common data) 3-5

3-2 Materials Shape Unit (MTR) 3-8

3-2-1 Setting unit data 3-8

3-2-2 Setting sequence data 3-8

3-3 Bar-Materials Machining Unit (BAR) 3-11

3-3-1 Setting unit data 3-11

3-3-2 Setting sequence data 3-15

3-4 Copy-Machining Unit (CPY) 3-20

3-4-1 Setting unit data 3-20

3-4-2 Setting sequence data 3-22

3-5 Corner-Machining Unit (CNR) 3-23

3-5-1 Setting unit data 3-23

3-5-2 Setting sequence data 3-24

3-6 Edge-Machining Unit (EDG) 3-25

3-6-1 Setting unit data 3-25

3-6-2 Setting sequence data 3-26

3-7 Threading Unit (THR) 3-28

1 MAZATROL PROGRAM CONFIGURATION

1-1 Program Configuration

MAZATROL programs are each made up of a set of data referred to as unit.
The following types of units are prepared for this NC equipment:

Common unit

Materials shape unit (MTR)

Bar-materials machining unit (BAR)
Copy-machining unit (CPY)
Corner-machining unit (CNR)
Edge-machining unit (EDG)
Threading unit (THR)
Grooving unit (GRV)
Drilling unit (DRL)
Tapping unit (TAP)
Manual program machining unit (MNP)

] Turning unit

M-code unit (M)
Measurement unit (MES)
Subprogram unit (SUB)

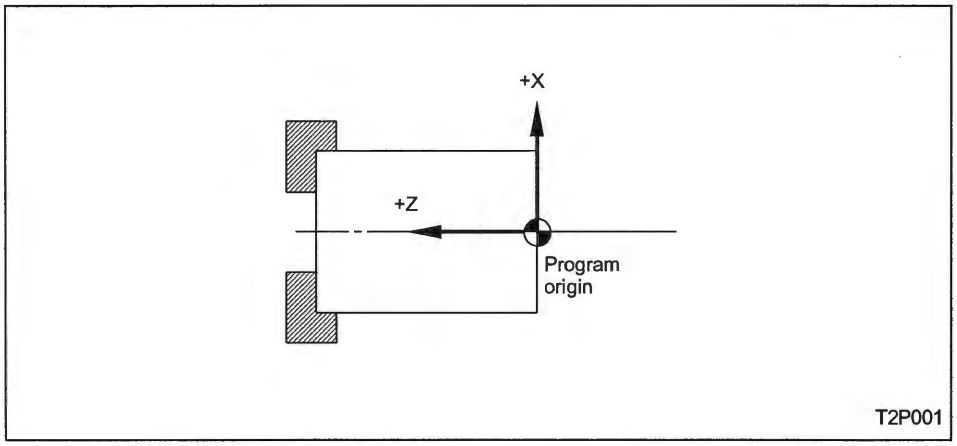
End unit (END)

Data to be set in the units listed above is classified into the following three major types:

1. Unit data
   Unit data must be set on the line of **UNo.** * (* : Unit number) of the **PROGRAM** display. The data consists of data on the type of machining, the sections to be machined, cutting conditions, machining tool numbers, etc.
2. Sequence data
   Sequence data must be set on the line of **SEQ** * (* : Sequence number) of the **PROGRAM** display. The data consists mainly of data used to define machining patterns.
3. TPC Data (Tool path control data)
   TPC data is the auxiliary data to be set on the TPC display. The data consists of tool path/tool change position adjustment data, M-codes, tool offset numbers, etc. Tool paths are automatically generated according to the data set on the **PROGRAM** display and various parameters. TPC data is intended to eliminate unnecessary paths by changing thus-generated tool paths on a unit-by-unit basis. Machining itself, therefore, will be executed even if TPC data is not set.

2 PROGRAM COORDINATE SYSTEM

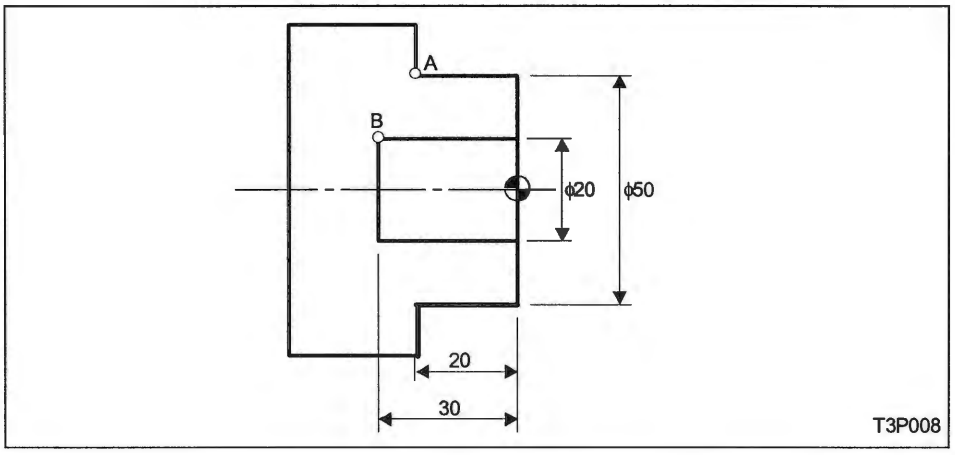
In general, machining dimensions on a drawing are indicated as the distances from a specific reference point. Likewise, within a program, a machining pattern is defined by setting the coordinates from a specific reference point. This reference point is referred to as the program origin and the coordinate system based on the program origin is referred to as the program coordinate system.
For MAZATROL programs, the following coordinate system is used to define machining patterns:



The program origin of X-Z-coordinates system can be set anywhere on the center line of the workpiece. Usually, however, the crossing point of the center line of the workpiece and its finishing edge surface should be taken as the program origin.

For MAZATROL programs, set X-coordinates as diameter data. That is, the workpiece diameter indicated on the drawing must be set as it is.

Example: For the workpiece shape shown in the diagram below:
The coordinates (x, z) of point A are (50, 20), and the coordinates (x, z) of point B (20, 30).



Note 1: For manual program machining units (MNP), milling manual program machining units (MMP) and edge-machining units (EDG), the direction of Z-axis is opposite to the one shown in the diagram above. See the relevant items in Chapter 3 for further details.

3 PROGRAM CREATION

Both the program data and sequence data within a MAZATROL program must be set on the **PROGRAM** display, and TPC data must be set on the **TPC** display. The TPC display is called up from the **PROGRAM** display.

This section first describes general procedures and precautions related to creating a MAZATROL program and then describes detailed procedures for setting each type of program data on a unit-by-unit basis.

Operating procedure for creating a MAZATROL program

(1) Select the **PROGRAM** display.

- Carry out the following operations to call up the **PROGRAM** display:

1) Press the display selector key.
You will then see the following main-display selection menu in the menu display area of your screen:

POSITION	SET UP INFO.	PROGRAM	TOOL DATA	CUTTING COND.	PARAM	DIAGNOS	DATA IN/OUT		DISPLAY MAP

2) Press the **PROGRAM** menu key.

- The program last selected will be displayed on the **PROGRAM** display and the current menu will change over to this one:

WORK No.	FIND	PROGRAM	HELP	TPC		LAYOUT	TOOL PATH		PROGRAM FILE

(2) Press the **WORK No.** menu key.

- The display of **WORK No.** becomes reversed and the work-Nos. listing window will be displayed.

* The work-Nos. listing window refers to a window that displays a list of work numbers of the programs that have already been registered in the NC equipment.

(3) Set the work number of the creating program.

- A work number is a number to be assigned to a program for identification of the program. Any number from 1 to 99999999 can be used as a work number.
- If a work number already registered in the NC unit is set, that program will be displayed on the screen. To create a new MAZATROL program, therefore, you must set a work number not used in other programs.
You can check the work-Nos. listing window or the **PROGRAM FILE** display to see which work numbers are not yet used.
- If you set a work number not used for the programs that have been registered in the NC unit, the current menu will change over to this one:

	*								
WORK No.	EIA/ISO PROGRAM	MAZATROL PROGRAM							

* The EIA/ISO programming function is optional.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (unit data line).

If you have selected a unit that consists of unit data and sequence data of multiple lines (unit MTR, BAR, CPY, THR or MNP):

(7)-1 Set data in each item on the unit data line.
- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (sequence data line).

(7)-2 Set data in each item on the sequence data line.
- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.

(7)-3 After you have set the entire sequence data, press the **SHAPE END** menu key.
- The line that immediately succeeds the last sequence data line will be displayed as a unit data line.
- For a unit that permits you to set more than one line of sequence data, you cannot select the next unit unless you carry out this operation (pressing the **SHAPE END** menu key).

(8) Select the units required for the intended machining operation by repeating steps (6) and (7) above (including steps (7)-1, (7)-2 and (7)-3), and then set data in each of the items displayed on the screen.
- A selectable unit differs according to the type of product to be machined. Select a unit in the most suitable order in accordance with your machining drawing, process sheet, etc. After unit selection, the program can be generated just by setting data as guided by messages.

(9) Set the end unit at the end of the program.
- Press the **END** menu key.
- Without the end unit, the program will not be regarded as a complete one. Therefore, you must set the end unit at the last line of the program.

(10) Set data in each item of the end unit.
- See the section "End Unit (END)" for details of the data to be set.

Note 1: If your NC unit has only standard functions, one MAZATROL program can contain a maximum of 99 units, including the common unit and the end unit. For units that allow you to set multiple lines of sequence data, up to a maximum of 25 lines of sequence data can be registered per unit (for manual program machining units only, up to a maximum of 250 lines of sequence data can be registered).

Note 2: The shape data that you have set can be checked for errors by calling up the **SHAPE CHECK** display while you are creating the program. See the Operating manual for details.

Note 3: For the following units, TPC data can be set as required:
Bar-materials machining unit (BAR)
Copy-machining unit (CPY)

3-1 Common Unit

The common unit is the first to be placed in a MAZATROL program, and always takes unit number 0.
Data that is set in this unit is referred to as common data, which becomes the base data for the entire program. When creating a MAZATROL program, therefore, you must first set data in this unit.

3-1-1 Setting unit data (common data)

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	RPM	FIN-X	FIN-Z	WORK FACE
0	•	•	•	•	•	•	•	•

• MAT

The following menu will be displayed when the cursor is placed at this item:

FC250	FCD450	S45C	SCM415	SUS440	A5052	SS400	AC4C		>>>
For USA									
Class #35	65-45-12	1045	8620	440	5052	1020	356.0		>>>
For UK									
GR260	500/7	080A42	655M13	431S29	5052	43A	LM25		>>>

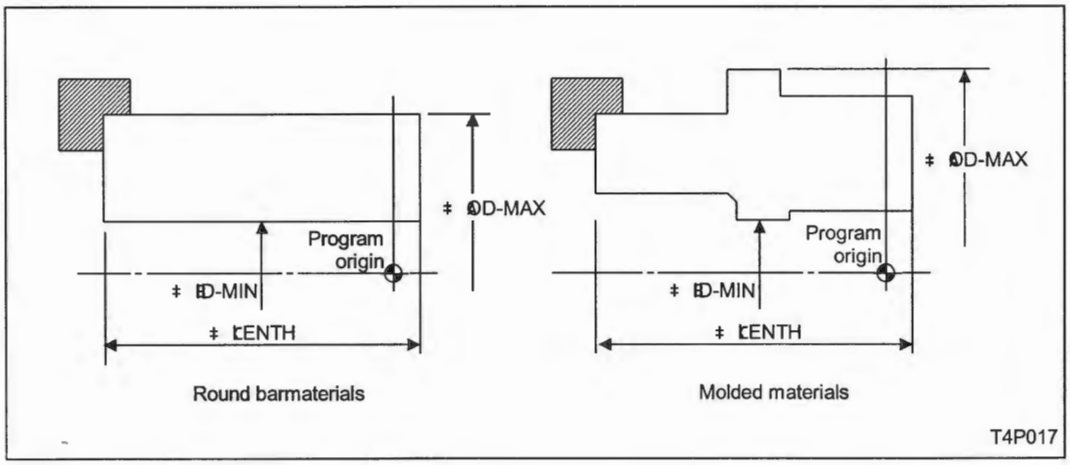
Note: The displayed menu differs according to the data set at WORK MAT item on the **CUTTING CONDITION (REGIST)** display.

From the menu, select the materials type of the workpiece to be machined.

If the workpiece to be machined is of a materials type other than those listed above, pre-register that materials type on the **CUTTING CONDITION (PERCENTAGE)** display. See the Operating manual for details.
The data of this item is referred to by the system during automatic setting of cutting conditions.

• OD-MAX, • ID-MIN, •LENGTH

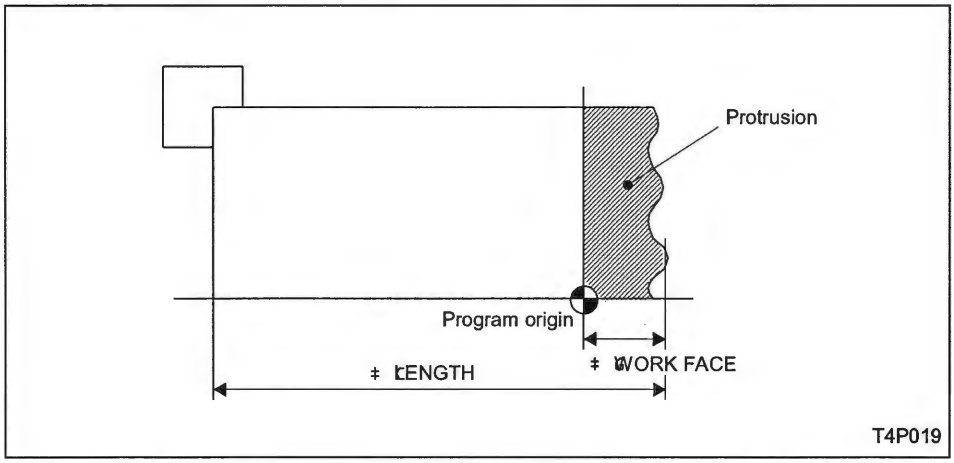
Set the maximum outside diameter, minimum inside diameter, and maximum length, respectively, of the workpiece.



- Set the workpiece length, including the edge protrusion (edge section to be cut), in item •.

• WORK FACE

Set the length of the workpiece edge protrusion in the Z-axis direction.



- The workpiece edge protrusion refers to a section to be cut during an edge-machining unit (EDG). For units other than edge-machining units, the protrusion is not regarded as part of the workpiece. Therefore, if the workpiece edge is to be cut (that is, if a value other than 0 is set for this item), an edge-machining unit must be selected before selecting a unit involving other machining operations.
  Either 0 or a plus value must always be set for this item.

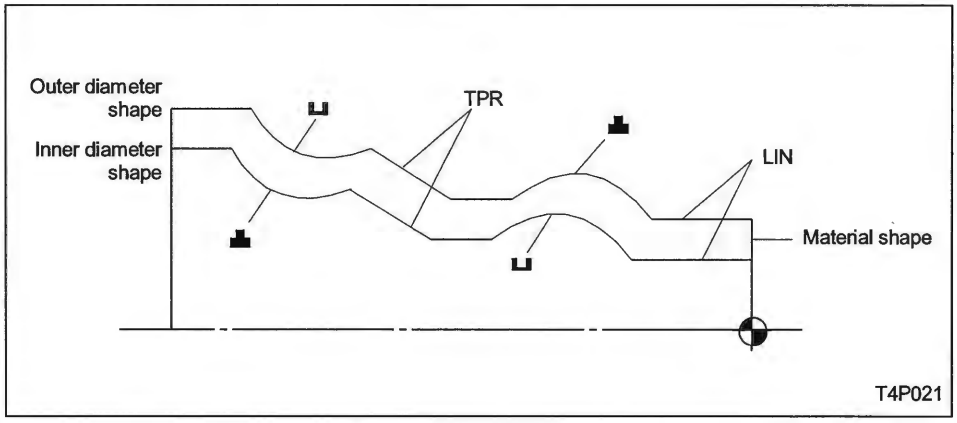
The data of the displayed menu denote the following shapes:

LIN: Line parallel to the center line of the workpiece

TPR : Line not parallel to the center line of the workpiece (Taper line)

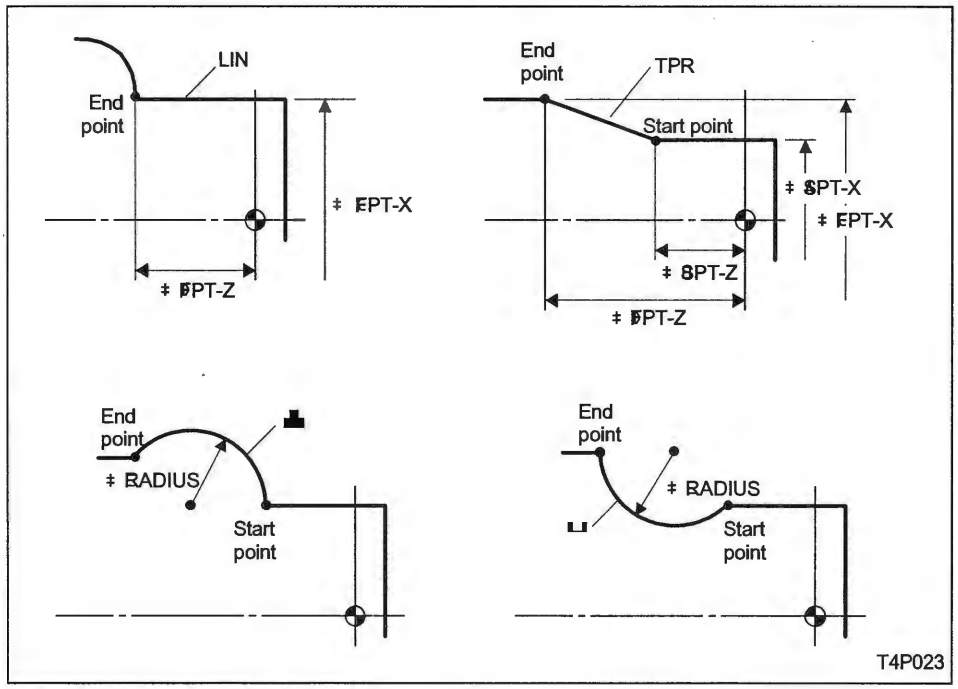
: Convex arc

: Concave arc



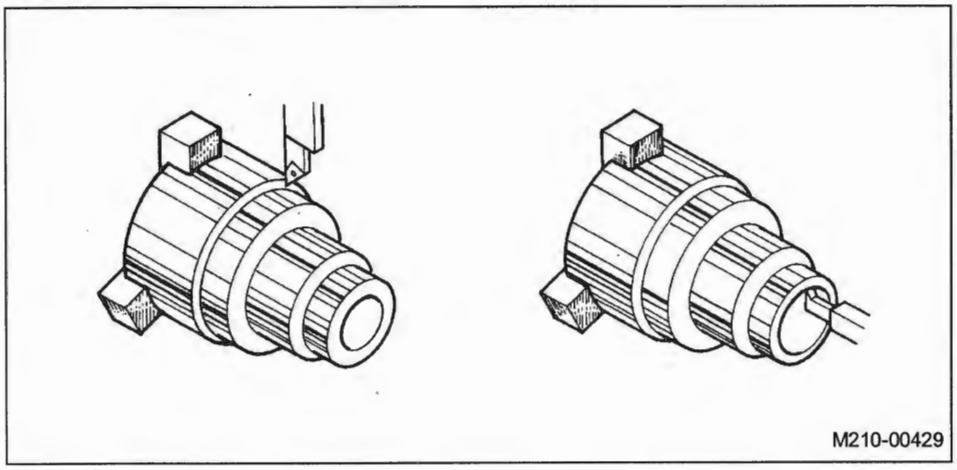
• SPT-X, • SPT-Z, • FPT-X, • FPT-Z, • RADIUS

Set the coordinates of the intended start point and end point of the shape you selected for item •. Also set the radius of the desired circle if you have selected ⌒ or ⌓.



3-3 Bar-Materials Machining Unit (BAR)

Select the bar-materials machining unit to lathe the outer peripheries, inner peripheries, front faces, or back faces of round-bar-materials using general-purpose cutting tools.



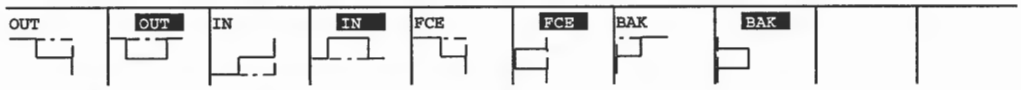
Press the **BAR** menu key to select this unit.

3-3-1 Setting unit data

UNo.	UNIT	#	DEP-X	CPT-Z	RV	FV	R-FEED	R-DEP.	R-TOOL	F-TOOL
*	BAR	•	•	•	•	•	•	•	•	•	•

• UNIT

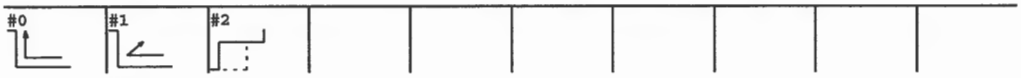
The following menu will be displayed when the cursor is placed at this item.



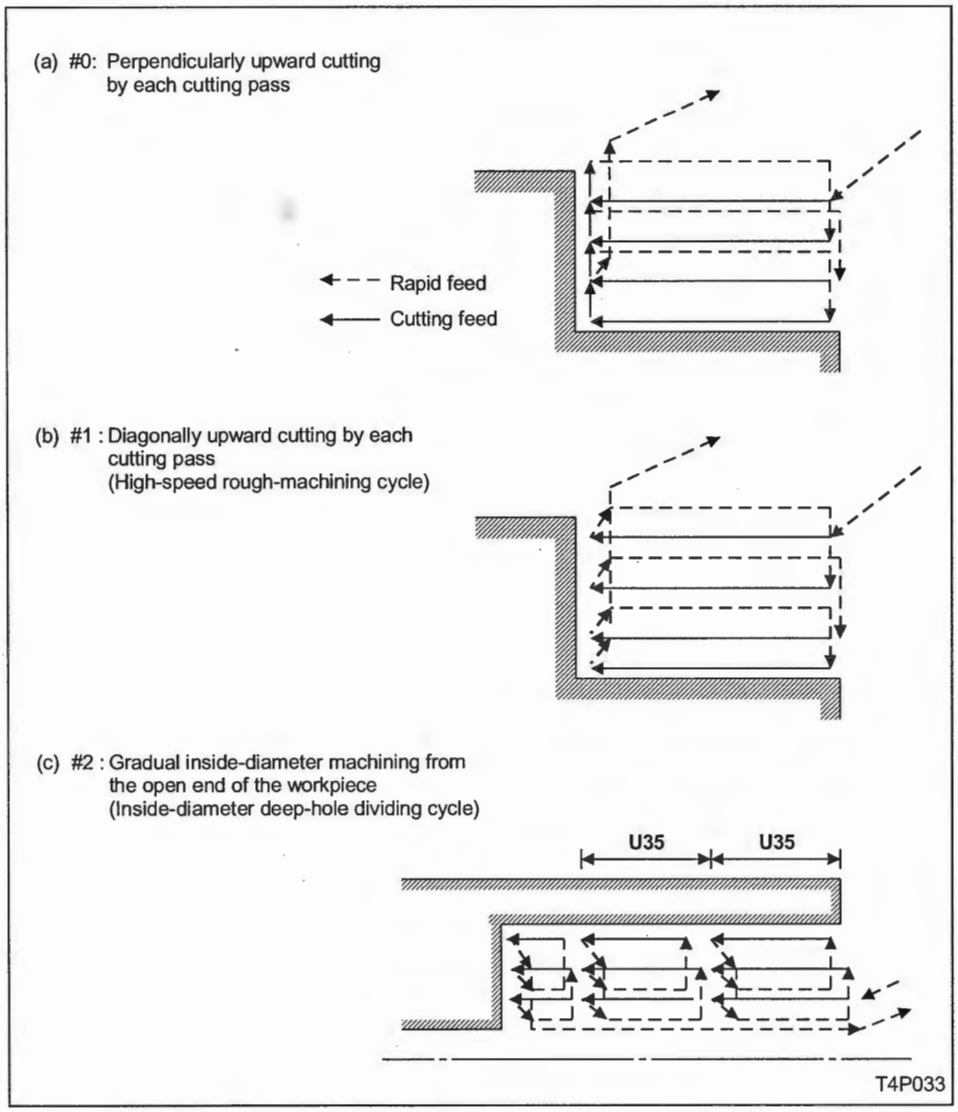
From the menu, select the section to be machined.

• #

The following menu will be displayed when the cursor is placed at this item.



Sekect a machining pattern from (a) through (c) above. The data of the displayed menu denote the following machining patterns:


(a) #0: Perpendicularly upward cutting by each cutting pass
Rapid feed
Cutting feed
(b) #1 : Diagonally upward cutting by each cutting pass (High-speed rough-machining cycle)
(c) #2 : Gradual inside-diameter machining from the open end of the workpiece (Inside-diameter deep-hole dividing cycle)
U35
U35
T4P033

- #2 machining can be used only when **IN** is selected for item •.
  The #2 cycle, however, cannot be selected to turn inside diameter for a shape of which the size increases with the depth.
  Chips may clog the hole bottom during conventional inside-diameter machining of deep holes. No such problems occur with this machining pattern (#2) since gradual cutting from the open end of a workpiece ensures highly efficient, automatic removal of chips.
  Use parameter **U35** to specify the depth of cut per pass.

• RV, • FV, • R-FEED, • R-DEP.

These four items denote the following data:

RV : Peripheral spindle speed during roughing
FV : Peripheral spindle speed during finishing
R-FEED : Feedrate per revolution during roughing
R-DEP. : Maximum depth of cut per pass during roughing (The data for the X-axis direction must be set using the radius value of the workpiece.)

Pressing the **AUTO SET** menu key with the cursor positioned at item • automatically sets NC-calculated data in the fields of items • through •.

Note: See "Automatic Cutting-Conditions Setting Function" for a description of base data and calculation expressions used for automatic setting of these four types of data.

• R-TOOL, • F-TOOL

Select a tool to be used for roughing in item • and for finishing in item •.
Tools can automatically be selected by pressing the **AUTO SET** menu key. See "Tool Auto-Setting" for further details.

Tools can be designated using the tool number method or tool code method (parameter setting). It is possible with the tool number method to assign suffixed (identifiers) by designating parameter. By adding a suffix, one tool can be handled as multiple tools.

- Only roughing will be executed if only an R-TOOL is designated.
- Only finishing will be executed if only an F-TOOL is designated.

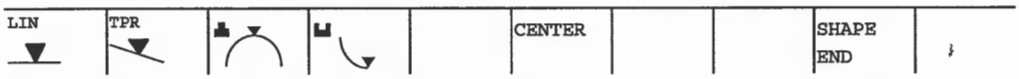
3-3-2 Setting sequence data

UNo.	UNIT	#	CPT-X	CPT-Z	RV	FV	R-FEED	R-DEP.	R-TOOL	F-TOOL
*	BAR ***	*	***	***	***	***	***	***	***	***

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	•	•	•	•	•	•	•	•	•

• SHP

The following menu will be displayed when the cursor is placed at this item.



(a) (b) (c) (d) (e) (f)

- Select a machining pattern from (a) through (d) above.
  The meaning of each data of the displayed menu is the same as for the materials shape unit (MTR).
- Select (e) to calculate the coordinates of the crossing point of a concave or convex section on the preceding SEQ line.
- Select (f) to proceed to the next unit after entire shape data have been entered.

• S-CNR

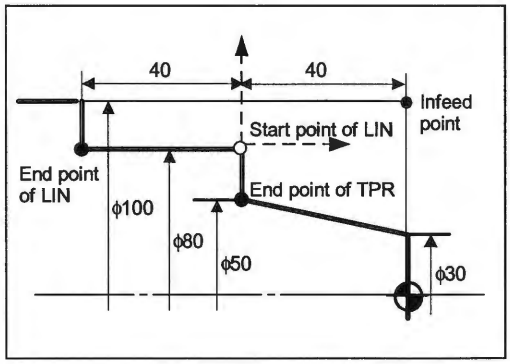
Set data for this item when C-chamfering (cornering) or R-chamfering (rounding) is to be done at the start point of the shape.

- If the selected shape type is LIN, the coordinates of the start point need not be set.
  The NC unit will auto-set those coordinates. A horizontal line will be drawn from the end point of LIN towards the infeed point, and the crossing point of this line and the line that is perpendicularly drawn from the end point of the preceding SEQ (or from the infeed point for an LIN as the first SEQ) will be set as the start point of the relevant LIN.

Example:

UNIT	CPT-X	CPT-Z
BAR OUT	100.	0.

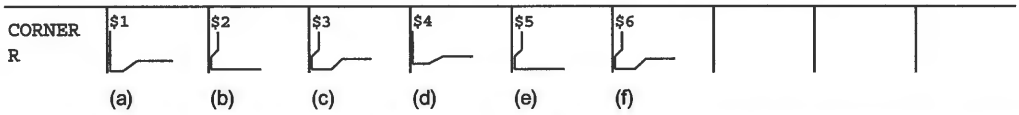
SHP	SPT-X	SPT-Z	FPT-X	FPT-Z
TPR	30.	0.	50.	40.
LIN	□Y	□Y	80.	80.



• F-CNR/•

Set data for this item when C-chamfering, R-chamfering, or pecking is to be done at the end point of the shape.

The following menu will be displayed when the cursor is placed at this item:



- The setting procedure for F-CNR is the same as for S-CNR (refer to the description and diagram for item • S-CNR).
- If pecking is to be done, select a pecking type from (a) through (f) above.
  Pecking types $4, $5 and $6 are the same as $1, $2 and $3, respectively. Use parameters **K33** to **K40** to select the dimensions of pecking. (See the Parameter List for further details.)
- Pecking can be done only if the following conditions are satisfied:
  1) During finishing.
  2) The selected shape and the next shape are linear and orthogonal.
  3) The cutting-edge angle and tip angle of the tool to be used satisfy the conditions listed in the table below.

2) Next, from the above menu, select one of the surface roughness codes indicated on the machining drawing. The above codes of the displayed menu denote the following levels of surface roughness:

• 1	• 2	•• 3	•• 4	••• 5	••• 6	••• 7	•••• 8	•••• 9	Surface roughness •µm•
100	50	25	12.5	6.3	3.2	1.6	0.8	0.4	
100-S	50-S	25-S	12-S	6-S	3-S	1.5-S	0.8-S	0.4-S	• finishing symbols
•	•	••	••	•••	•••	•••	••••	••••	

The finishing feedrate is calculated from the following expression automatically:

$$F = \sqrt{\frac{8R\mu}{1000}}$$

F: Finishing feedrate (mm/rev)
R: Radius of tool nose (mm)
µ: Surface roughness (µm)

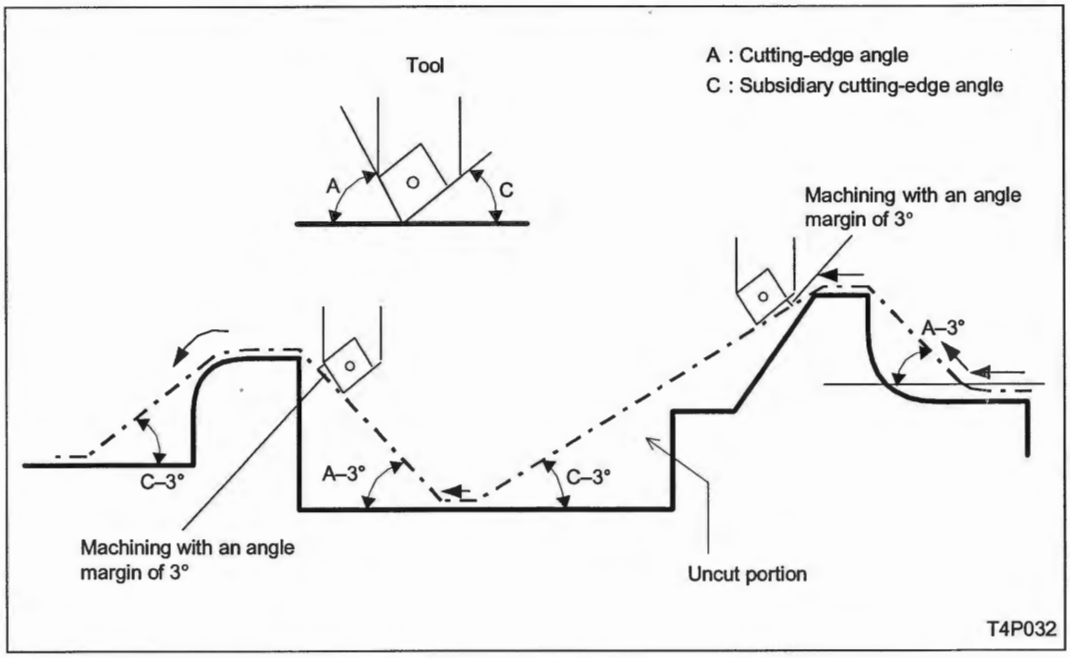
- If a feedrate is to be directly set:
  After pressing the **FEEDRATE/rev** menu key, set the desired value.

Note 1: During sequences having no data set for this item, finishing is done at the feedrate that was set for unit data item •, R-FEED.

Note 2: The same value will automatically be set here if the preceding sequence has a set data of RGH.

<Precautions for BAR unit>

Some parts may remain uncut because of the tool shape.
For a tool having a cutting-edge angle of A and a subsidiary cutting-edge angle of C, uncut portions occur at an angle of A – 3° in the machining direction and an angle of C – 3° in an opposite direction because machining will occur generally with an angle margin of 3°.

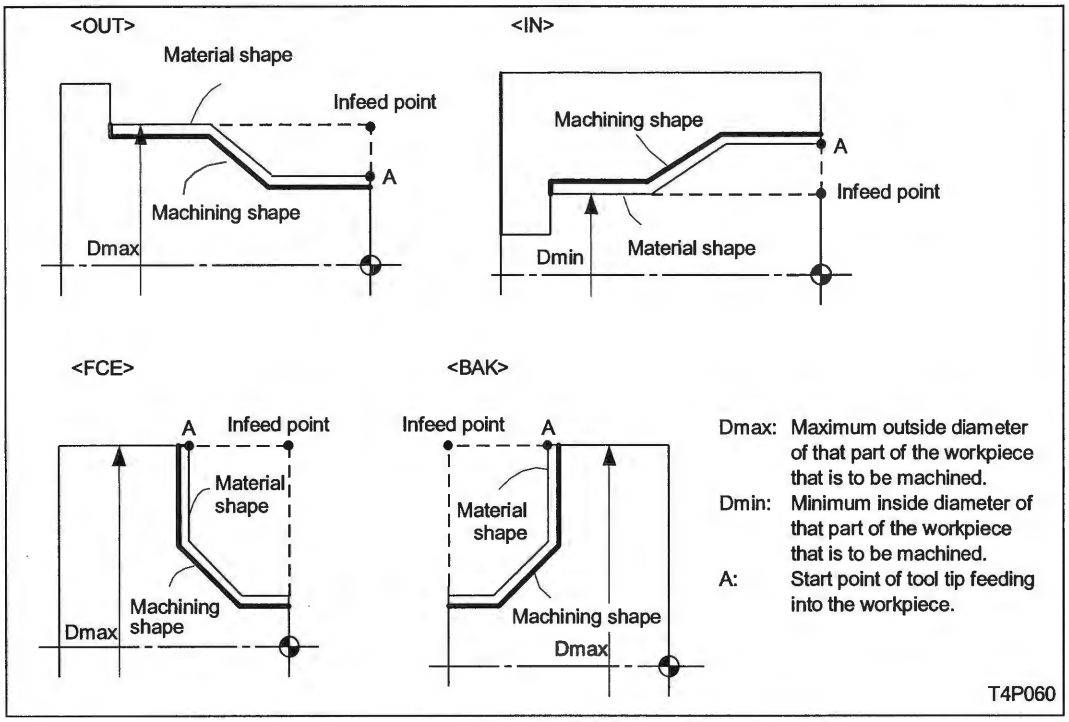


* The description given above also applies to CPY unit.

• CPT-X, • CPT-Z

Set the X- and Z-coordinates of the desired infeed point.

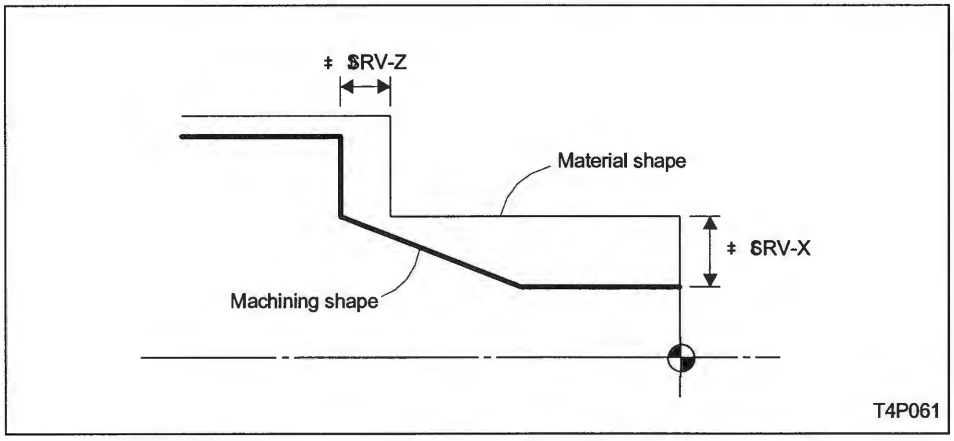
In general, the infeed point means the start point of tool tip feeding into a workpiece. For copy-machining units, however, the points shown in the diagrams below are infeed points.



• SRV-X, • SRV-Z

Set the maximum removal allowances in the X-axis and Z-axis directions (removal allowances for the section that is to be cut most deeply).

The removal allowance in the X-axis direction must be set with the radius value (half the workpiece thickness).



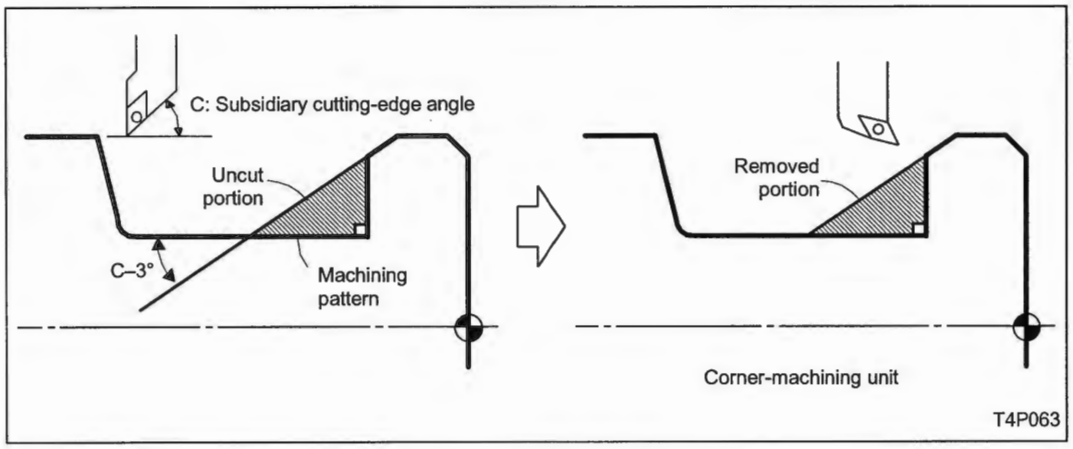
• RV, • FV, • R-FEED, • R-DEP.

These items have the same meanings as for BAR unit.

See "Bar-Materials Machining Unit (BAR)" for further details.

3-5 Corner-Machining Unit (CNR)

Part of the corners of a workpiece may remain uncut because of the particular tool shape for the bar-materials machining unit (BAR) or the copy-machining unit (CPY). Select the corner-machining unit (CNR) when uncut portions are to be removed to make all corners right-angled.



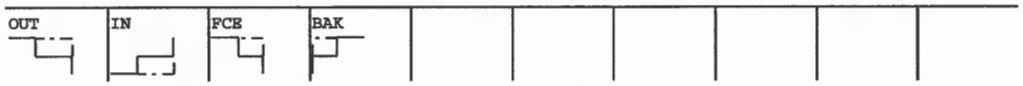
Press the **CNR** menu key to select this unit.

3-5-1 Setting unit data

UNo.	UNIT	#	RV	FV	R-FEED	R-DEP.	R-TOOL	F-TOOL
*	CNR •	•	•	•	•	•	•	•

• UNIT

The following menu will be displayed when the cursor is placed at this item.



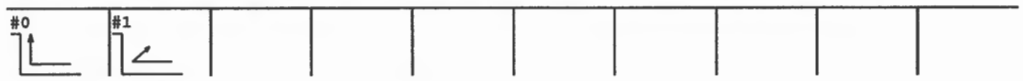
From the menu, select the section to be machined.

Sections to be machined that correspond to the data of the displayed menu are as follows.

OUT: Uncut portion on outer periphery
IN : Uncut portion on inner periphery
FCE: Uncut portion on front face
BAK: Uncut portion on back face

• #

The following menu will be displayed when the cursor is placed at this item.



(a) (b)

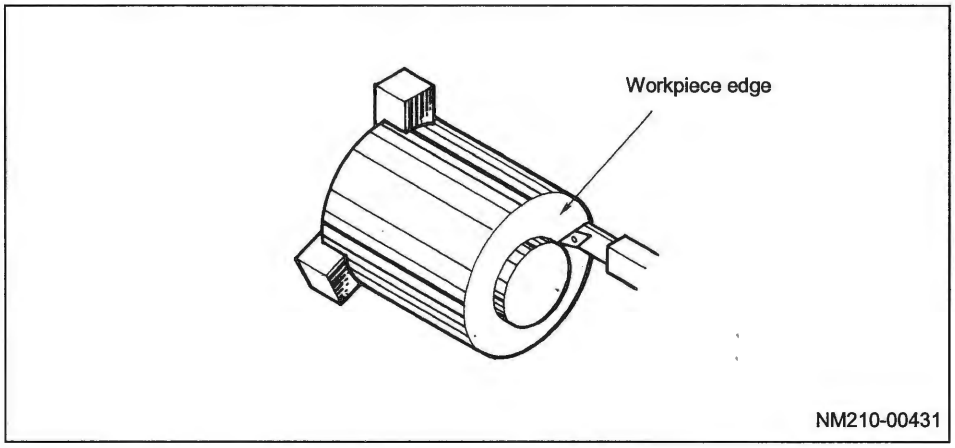
Select a rough-machining pattern from (a) or (b) above.
See the description of the relevant item for BAR unit.

• RV, • FV, • R-FEED, • R-DEP.

These items have the same meanings as for BAR unit.
See "Bar-Materials Machining Unit (BAR)" for further details.

3-6 Edge-Machining Unit (EDG)

Select the edge-machining unit (EDG) when chipping off any protrusions of the workpiece edges (front face or back face).



Press the **EDG** menu key to select this unit.

3-6-1 Setting unit data

UNo.	UNIT	RV	FV	R-FEED	R-DEP.	R-TOOL	F-TOOL
*	EDG •	•	•	•	•	•	•

•UNIT

Select FCE or BAK to machine the front face or back face of the workpiece, respectively.
Select FCE for the case shown above.

Note: The **BAK** menu item cannot be selected for some machine specifications.

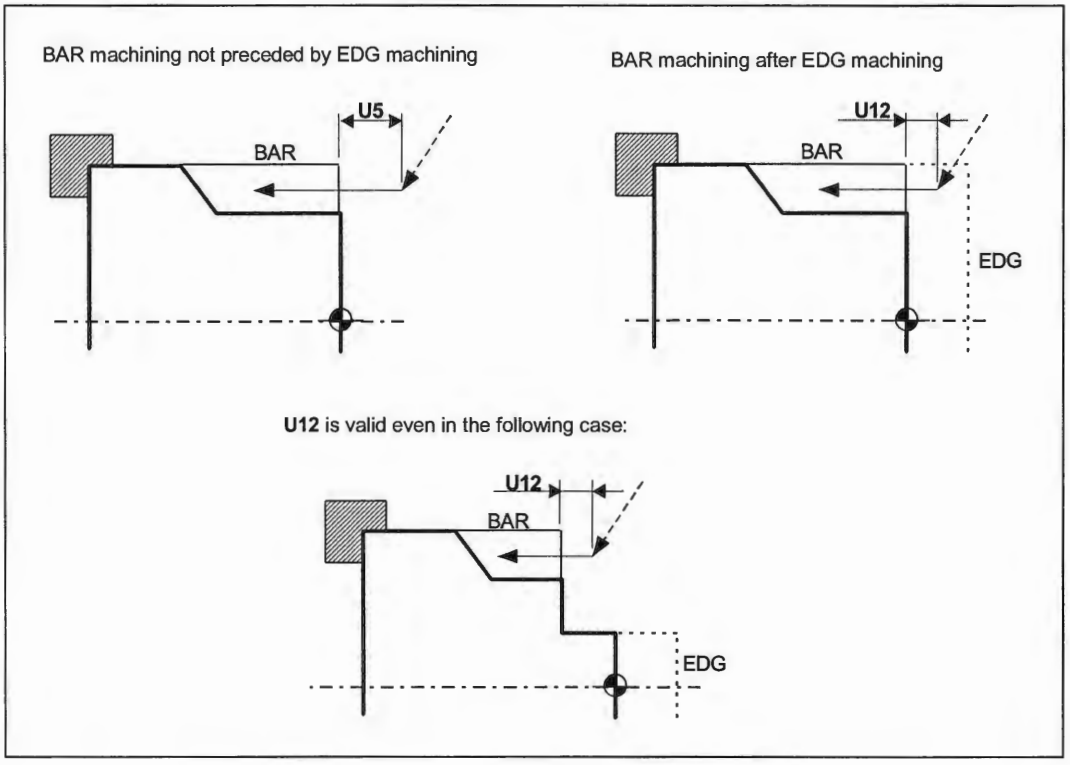
• RV, • FV, • R-FEED, • R-DEP.

These items have the same meanings as for BAR unit.
See "Bar-Materials Machining Unit (BAR)" for further details.

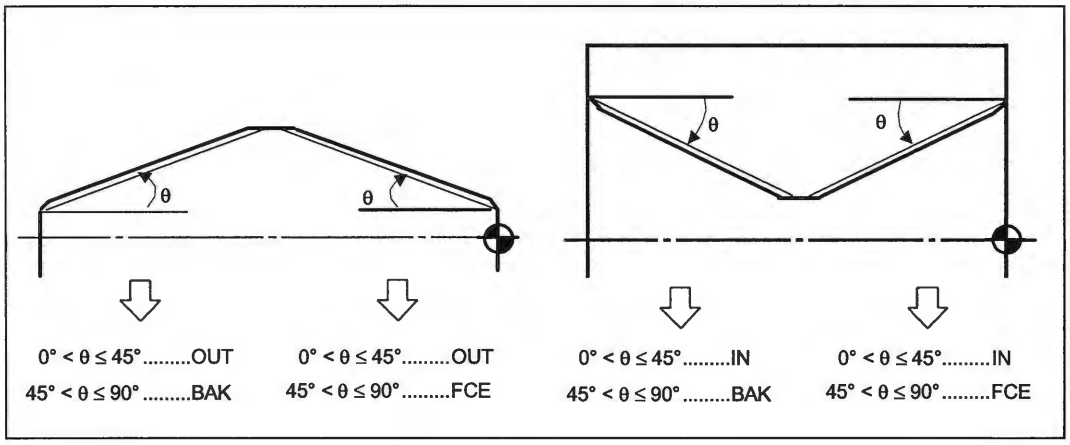
• R-TOOL, • F-TOOL

Select tools you want to use for rough-machining and finish-machining, respectively.
See the relevant items in "Bar-Materials Machining Unit (BAR)" for further details.

Note: The tool path for BAR and CPY units is calculated in general with the end-face clearance set in parameter **U5**.
For a BAR or CPY unit preceded by an EDG unit, in particular, the value of parameter **U12** is used as the end-face clearance.



- For taper threading, select the appropriate machining section as follows according to the desired taper angle:



• #

The following menu will be displayed when the cursor is placed at this item:

#0	#1	#2	#0	#1	#2				
STANDARD	CONST. DEPTH	CONST. AREA	STANDARD	CONST. DEPTH	CONST. AREA				

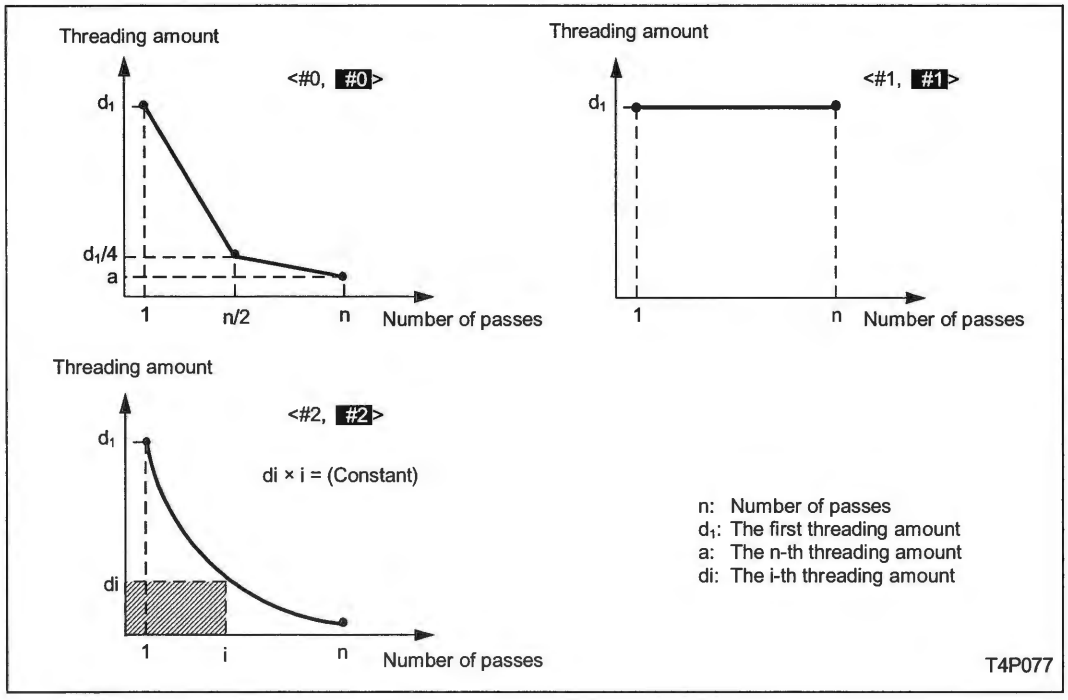
From the menu, select the threading pattern to be machined.

Patterns to be machined that correspond to the data of the displayed menu are as follows.

#0, #0 : Standard patterns
The threading amount gradually decreases as the number of passes increases.

#1, #1 : Constant-threading pattern
The threading amount always stays constant, irrespective of the number of passes.

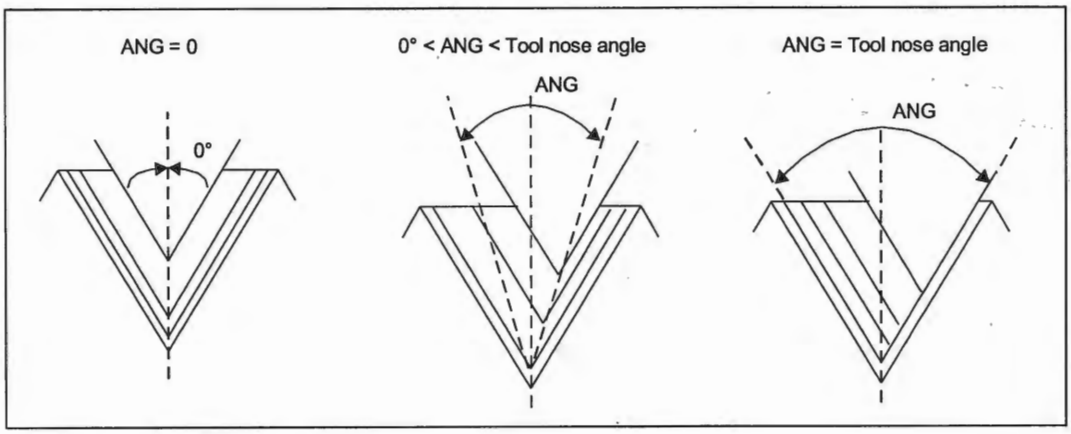
#2, #2 : Area-constant pattern
The threading amount is inversely proportional to the number of passes.



• ANG

Set a threading angle.

- Usually, set a several degrees smaller value than the nose angle of the tool.



• MULTI

Set the desired number of threads.

• HGT, •NUM., •C-SP, • DEPTH

These items denote the following data:

HGT : Threading height
NUM. : Number of times of threading
C-SP : Spindle peripheral speed during threading
DEPTH : The first threading amount (you must set the radius value of the workpiece as the amount of threading in the X-axis direction)

Pressing the **AUTO SET** menu key with the cursor at item • will automatically set data into items • through •.

Note 1: See "Automatic Cutting-Conditions Setting Function" for a description of the base data and calculation expressions used for these four types of data.

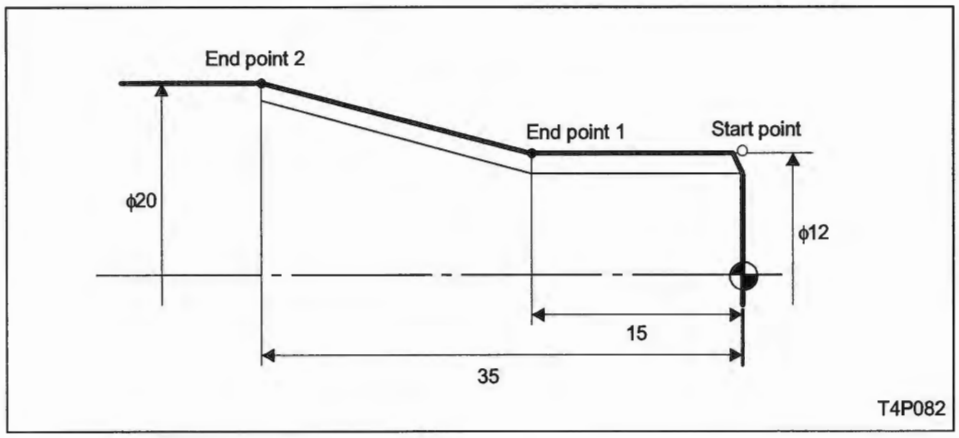
Note 2: It is not necessary to set data in the item displayed a • mark, for it denotes the data that is automatically determined jointly by the cutting pattern designated in item •, HGT, NUMBER and DEPTH.

Note 3: The NUMBER data must be an integral number equal to or higher than 3.

• TOOL

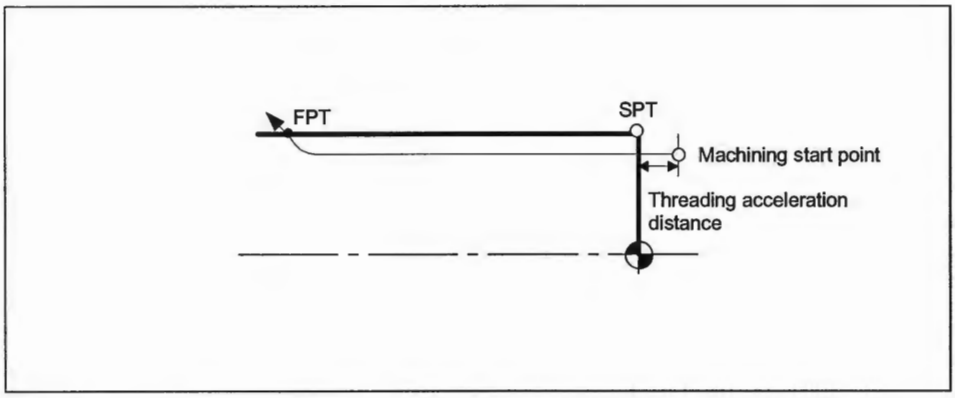
Select a threading tool you want to use.
For the threading unit R-TOOL/F-TOOL distinction is not drawn.

Note 1: The continuous threading pattern shown in the diagram below can be generated by setting multiple lines of sequence data.
In that case, the coordinates of the second and subsequent start points do not need be set (items • and • will be marked with •).



UNo.	UNIT	#	CHAMF	LEAD	ANG	MULTI	HGT
*	THR OUT	*	***	***	***	***	***
SEQ			SPT-X	SPT-Z	FPT-X	FPT-Z	
1			12.	0.	12.	15.	
2			•	•	20.	35.	

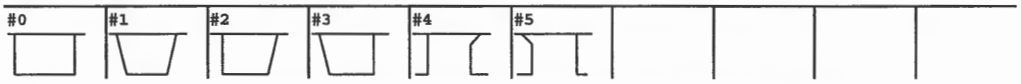
Note 2: Threading action begins at a position that is away from the start point specified in the program by the acceleration distance. Before carrying out a threading operation, therefore, check for possible interference with the tailstock or the workpiece during threading.



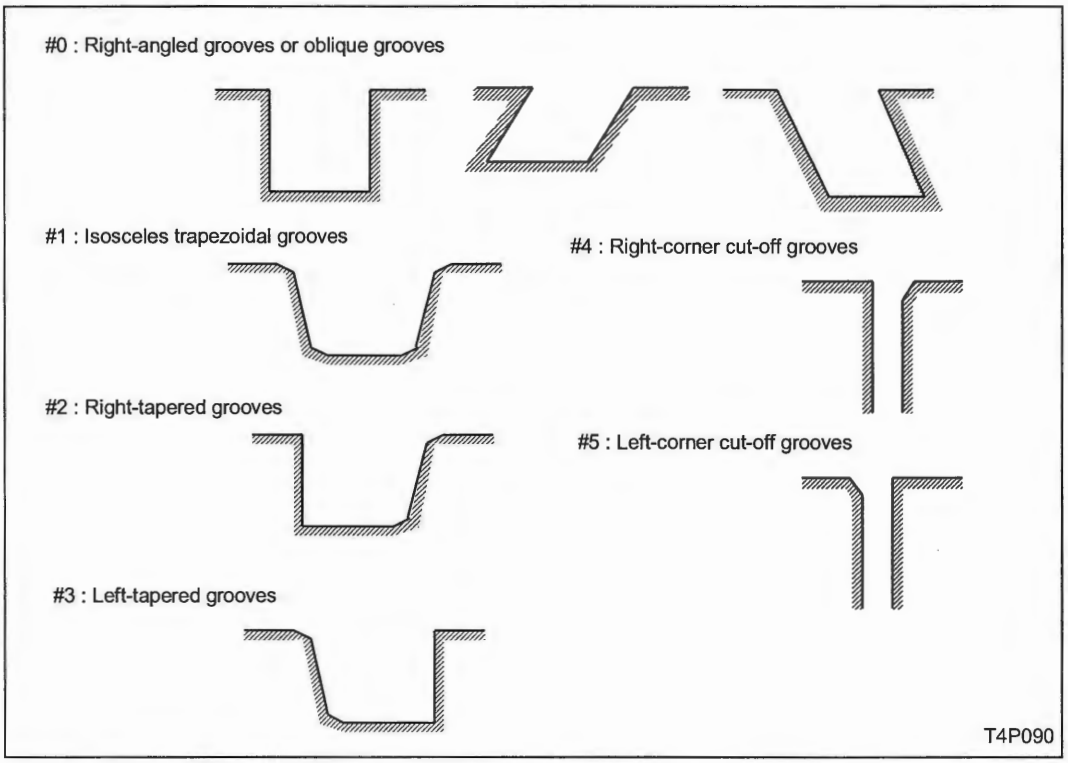
Note 3: The actuation of feed-hold function during a pass of threading will not interrupt the machine operation until the chamfering at the end point of threading has been completed.

• #

The following menu will be displayed when the cursor is placed at this item.



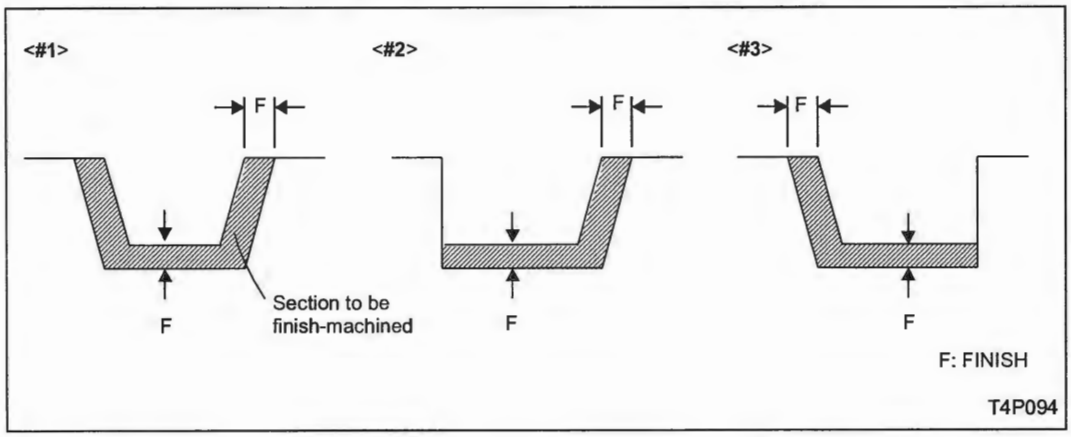
Select a grooving pattern from the menu. The data of the displayed menu denote the following grooving patterns:


#0 : Right-angled grooves or oblique grooves
#1 : Isosceles trapezoidal grooves
#4 : Right-corner cut-off grooves
#2 : Right-tapered grooves
#5 : Left-corner cut-off grooves
#3 : Left-tapered grooves
T4P090

- The OUT must be selected in item • above if you select grooving pattern #4 or #5 (Cutting off). For patterns #4 and #5, the MAZATROL T32 change mode or the new cycle mode can be selected by a parameter setting (**U56**).

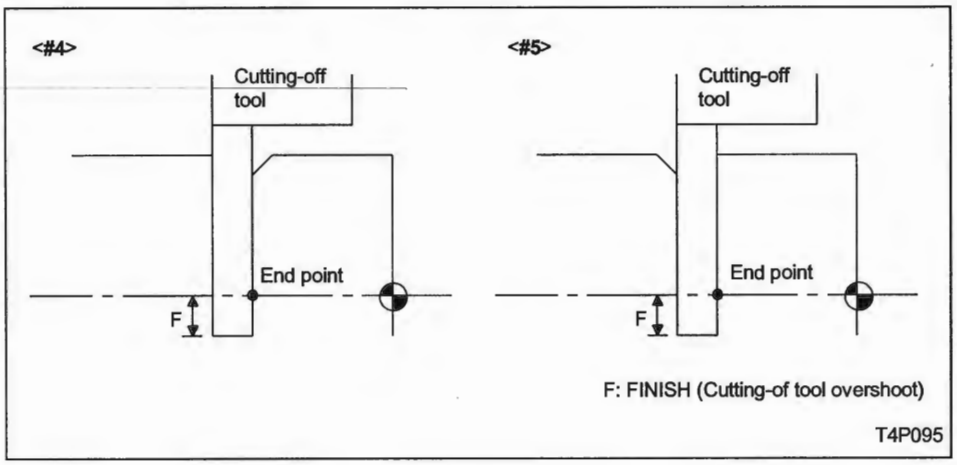
• FINISH

- No data can be set if you have selected grooving pattern #0.
- Set a finish-machining removal allowance if you have selected grooving pattern #1, #2 or #3.



Note: No finishing allowance will be provided to right-angled walls if you have selected grooving pattern #2 or #3.

- Set a cutting-off tool overshoot if you have selected grooving pattern #4 or #5.



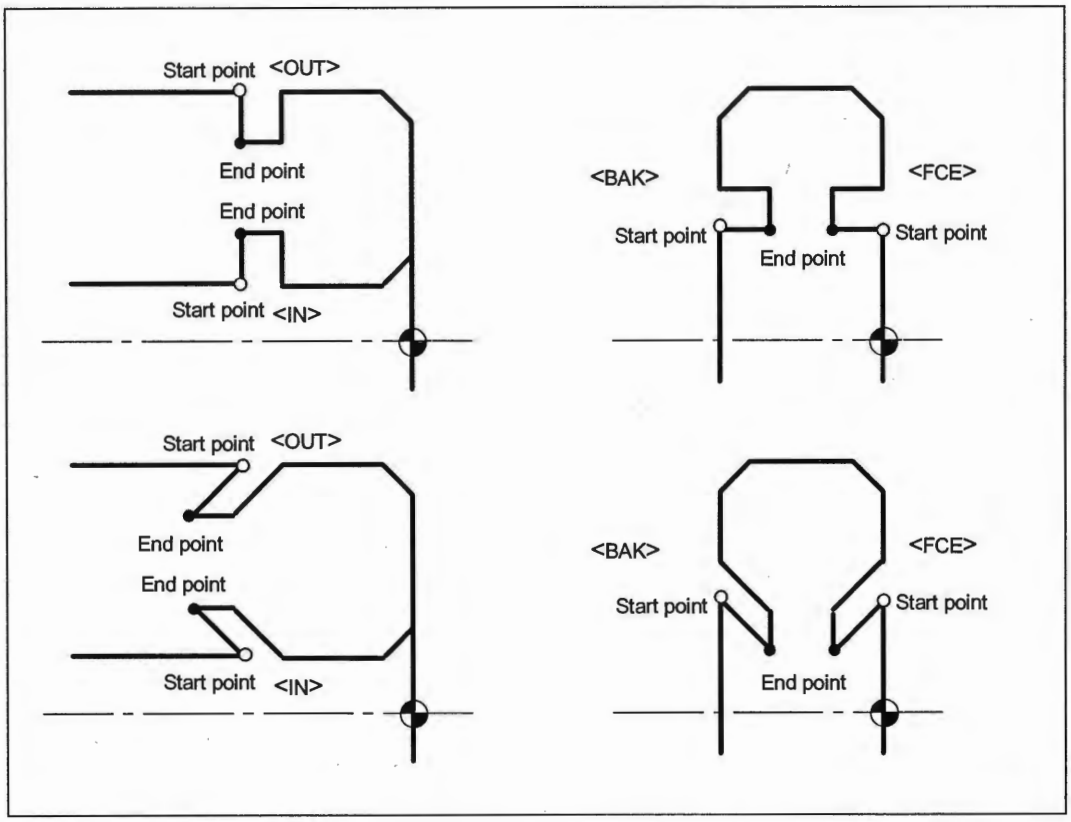
• RV, • FV, • FEED, • DEP.

- For each grooving pattern, set data into these items as follows:

Pattern	RV	FV	FEED	DEP.
#0	•	Peripheral speed during grooving	Feedrate during grooving (Seq. data RGH ineffective)	Max. depth of cut per pass (Designate in radius for OUT or IN)
#1	Peripheral speed during roughing	Peripheral speed during finishing	Feedrate during roughing	
#2 or #3				
#4 or #5 (T32 mode)	•	Peripheral speed during grooving	Feedrate during grooving (Seq. data RGH effective for parameterized cutting-off area)	Max. depth of cut per pass (Designate in radius; without pecking if 0 is set)
#4 or #5 (New mode)		Number of revolutions during grooving (*)	Starting feedrate for grooving (**)	

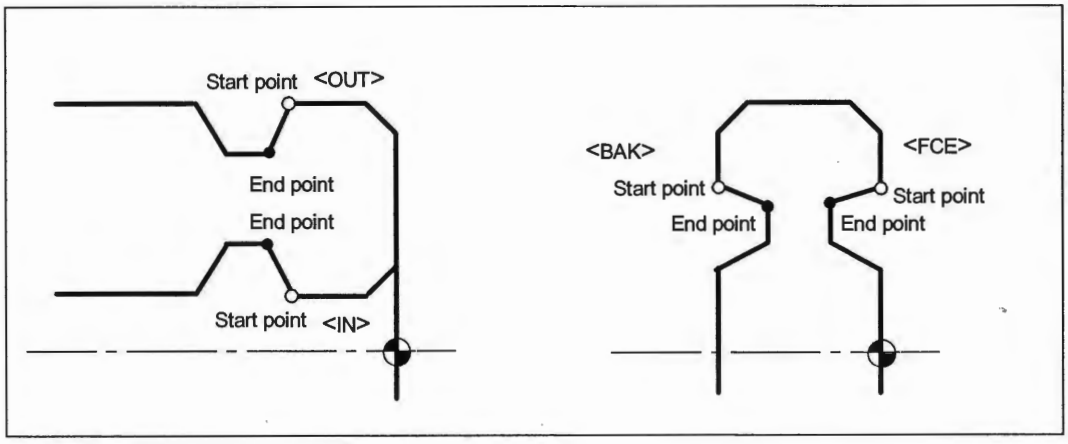
* The section from the starting point of machining to the parameter-designated cutting-off area is machined at the rotational speed designated here.

** The feedrate is reduced in several parameter-designated steps to the value of sequence data RGH.



For pattern #0, oblique grooves can be machined by setting the start and end points as shown above.

- For grooving pattern #1



Note 2: If multiple grooves of the same shape are to be machined (according to the setting of unit data "No."), set the coordinates of the start point and end point of grooving of the first groove.

Note 3: If grooving pattern #4 or #5 is selected, no data setting is required for **FPT-Z.**

• CNR

The data for the ending corner is only effective for patterns #1, #2, and #3.
See the description and the diagram for item •, S-CNR, for further details.

• ANG

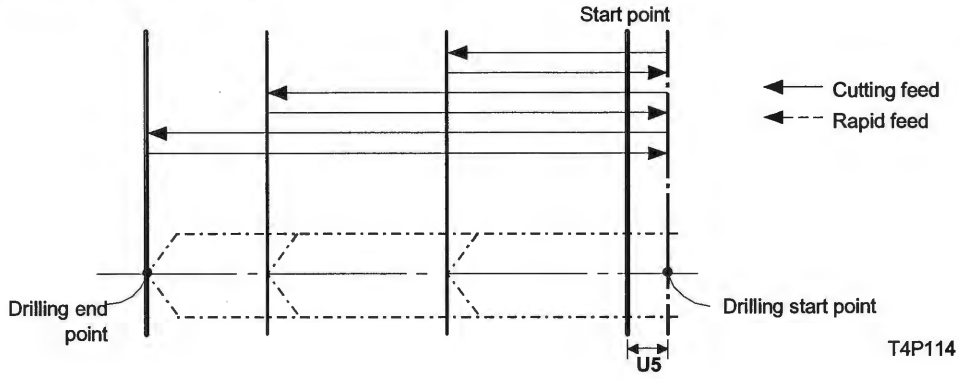
Set a taper angle if you have typed the question mark "?" in one of the four items from • to • above.
See "Automatic Crossing-Point Calculation Function" for details of setting a taper angle.

• RGH

For each grooving pattern, set data into this item as follows:

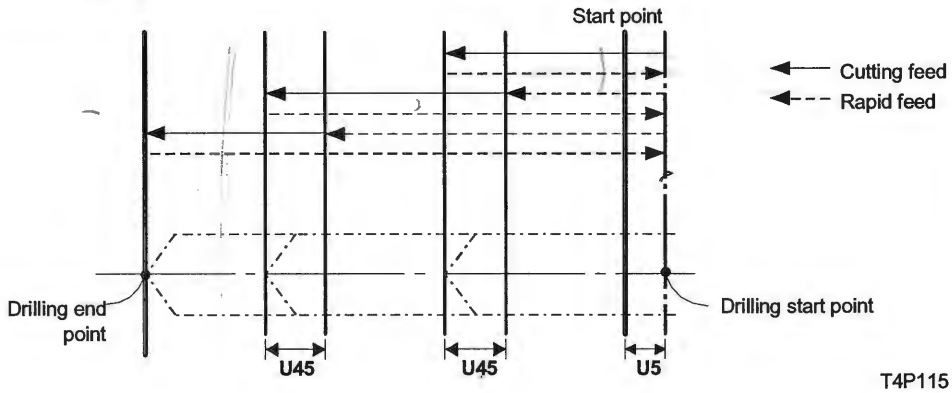
Pattern	Description of RGH data
#0	Invalid. (Set the feedrate in item FEED of unit data.)
#1, #2, #3	Set the feedrate during finishing. (Finishing will also be executed at the FEED rate of unit data if no data is designated here.)
#4 and #5 (both in T32 and New mode)	Set the feedrate for the cutting-off area. (Cutting-off will be executed at the half of the FEED rate of unit data if no data is designated here.)

#0, **#0**: Conventional type of drilling cycle
The drill returns to the drilling start point at a cutting feedrate after each infeed operation has been completed.



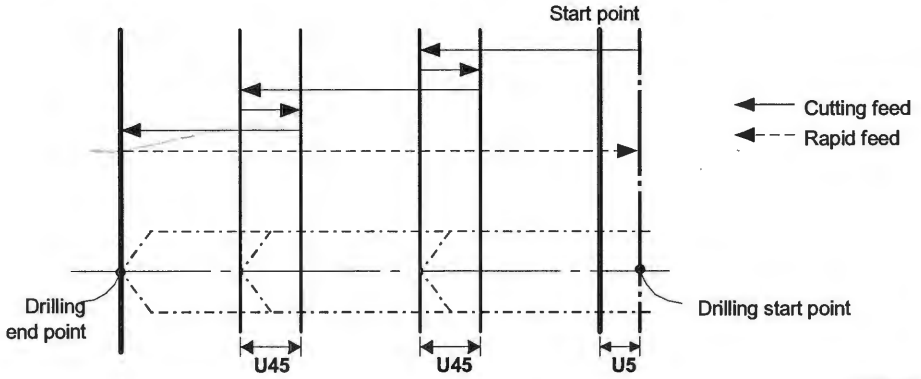
T4P114

#1, **#1**: Deep-hole drilling cycle
The drill returns to the drilling start point at a rapid feedrate after each infeed operation has been completed.



T4P115

#2, **#2**: High-speed deep-hole drilling cycle
The drill returns through the specified distance (data specified in parameter **U45**) at a cutting feedrate after each infeed operation has been completed.



T4P116

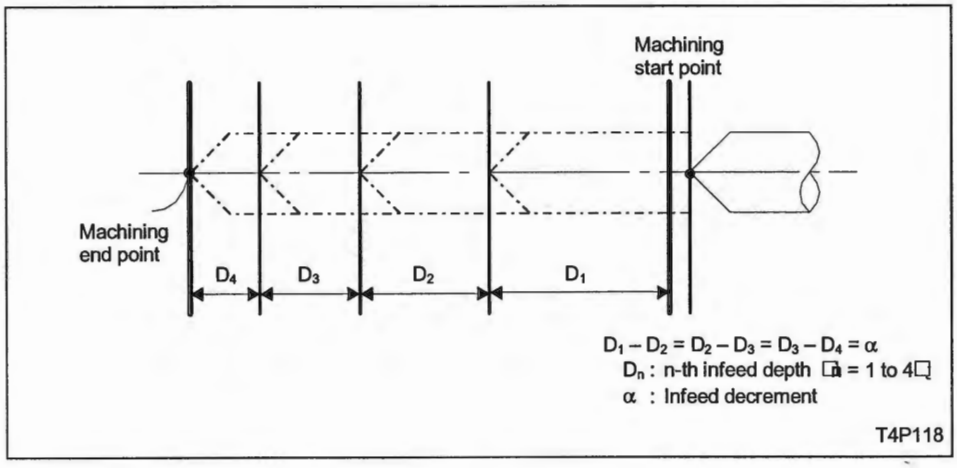
• DEP-1, • DEP-2, • DEP-3, •C-SP, • FEED

You can automatically set data in these items.

- When machining pattern other than #3 or #3 is selected.
  Pressing the **AUTO SET** menu key with the cursor at item • and setting a tool will automatically set the data that has been calculated by the NC unit into items • through •.
- When machining pattern #3 or #3 is selected.
  Items • through • will be marked with •. (Data cannot be set.)

Any desired data can be set in these items, and automatically set data can be changed. The data items denote the following data:

DEP-1 : First infeed depth
DEP-2 : Infeed decrement
DEP-3 : Minimum infeed amount



Taking the infeed decrement as α, one can calculate the n-th infeed amount, D_n ($n \geq 2$), as follows:

$D_n = D_{n-1} - \alpha = D_1 - \alpha \bullet n - 1 \bullet$

However, if $D_{n-1} - \alpha \leq D_{min}$ (D_{min}: minimum infeed amount), then

$D_n \bullet = D_{n+1} = D_{n+2} = \bullet\bullet\bullet\bullet\bullet\bullet\bullet = D_{min}$

C-SP : Spindle peripheral speed
FEED : Feedrate per revolution

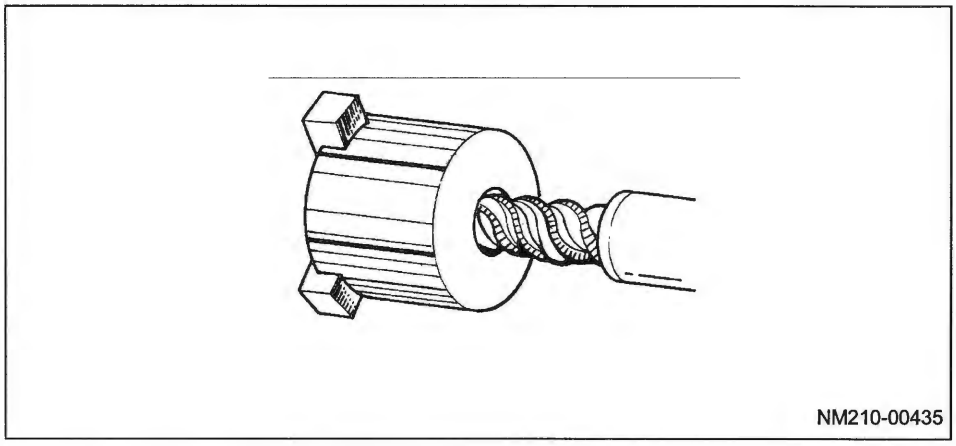
Note: See "Automatic Cutting-Conditions Setting Function" for a description of the base data and calculation expressions used for automatic setting of these five types of data.

• TOOL

Select a drill to be used. See the relevant items in "Bar-Materials Machining Unit (BAR)" for tool selection method. For drilling unit, R-TOOL/F-TOOL distinction is not drawn. The NC regards the drill as an R-tool.

3-10 Tapping Unit (TAP)

Select the tapping unit when holes in the middle of a workpiece are to be tapped.



NM210-00435

Press the **TAP** menu key to select this unit.

Note: To perform synchronous tapping (option), specify the required tool as **FIXED TAP** by menu selection on the **TOOL DATA** display.

3-10-1 Setting unit data

UNo.	UNIT	NOM-DIA	PITCH	C-SP	TOOL
*	TAP •	•	•	•	•

•UNIT

Select **FCE** or **BAK** to tap the workpiece from the right side or left side, respectively.
Select **FCE** for the case shown above.

Note: The **BAK** menu item cannot be selected for some specifications.

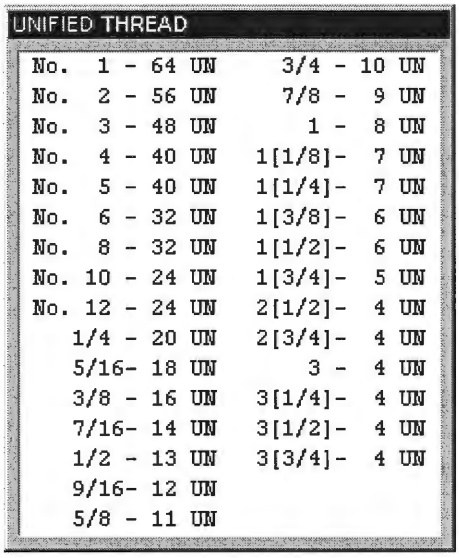
• NOM-DIA

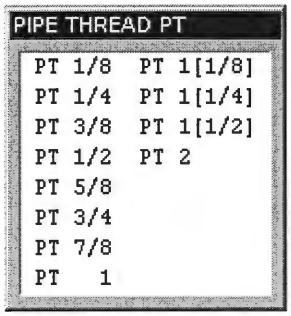
The following menu will be displayed when the cursor is placed at this item.

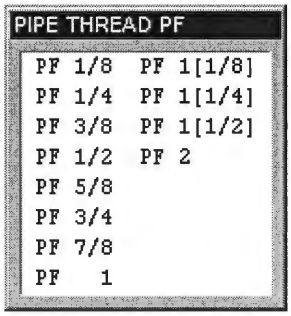
METRIC THRD (M)	UNFY THRD (UN)	PIPE THRD (PT)	PIPE THRD (PF)	PIPE THRD (PS)				OTHER	
(a)	(b)	(c)	(d)	(e)				(f)	

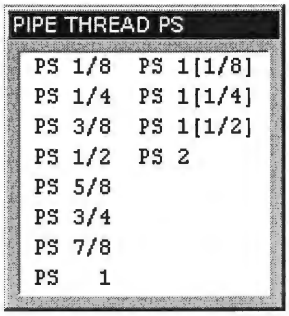
From (a) through (f) above, select the type of threads to be tapped. Then, set the nominal diameter of the threads.

- You can also select nominal diameter of the unified or pipe thread type on the window of the specified thread that will be displayed by pressing the **NOM-DIA SELECT** menu key.









Select the desired nominal diameter using the cursor keys and press the input key .
Then, the selected nominal diameter of the thread is set.

• PITCH, •C-SP

You can automatically set data in these items.
Pressing the **AUTO SET** menu key with the cursor at item • and setting a tool will automatically set the data that has been calculated by the NC unit into • and •.

Any desired data can be set in these items, and their automatic data settings can be changed.

The items denote the following data:

PITCH: Pitch of the threads to be tapped
(Pitch of the tapping tool to be used)

C-SP : Spindle peripheral speed during tapping

Note 1: The pitch is automatically set from the data that was set in item • above. See "Automatic Cutting-Conditions Setting Function" for a description of the base data and calculation expressions used for automatic setting of the peripheral speed.

Note 2: Automatic data setting is impossible if the thread type you selected in item • above is **OTHER.**

3-11 Manual Program Machining Unit (MNP)

The manual program machining unit complements the turning units described so far (BAR, CPY, CNR, EDG, THR, GRV, DRL and TAP).
These turning units have respective tool paths automatically generated according to the unit data and sequence data you have set, whereas the manual program machining unit requires user setting of its tool path.
Select this unit if a machining type or machine action that cannot be programmed in usual machining units is required, or if it is likely to be more convenient to directly set a tool path.
Press the **MANUAL PROGRAM** menu key to select this unit.

3-11-1 Setting unit data

UNo.	UNIT	CHANGE-PT	GEAR	TOOL
*	MNP	•	•	•

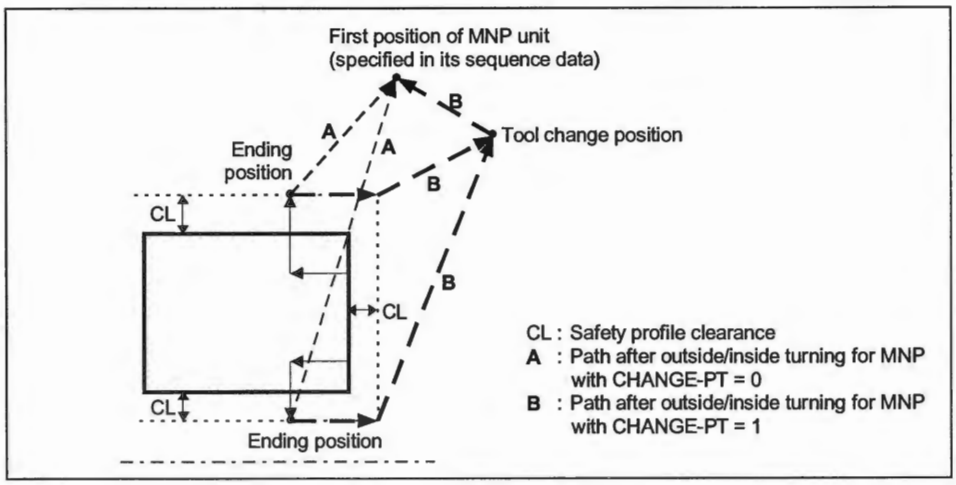
• CHANGE-PT

Specify whether you want to return the tool to a predetermined tool change position before executing the manual program machining unit.

- Set **1** if tool return is required.
- Set **0** if tool return is not required.

Setting **1** in this item returns a turret to predetermined tool change position, that is, the position previously specified using parameter **P17**, after the previous unit ends, and then the tool to be used for the manual program machining unit is indexed at that position.
Setting **0** in this item causes tool change to occur at the end position of the previous machining unit.
Before setting **0**, therefore, you must make sure that the tool does not contact the workpiece during tool change.

Note: The figure below illustrates the tool motion in the transition from an outside or inside turning unit to the MNP unit.
To avoid the interference on path "A" from the ending position of an inside turning, select path "B" (return to tool change position) by setting "CHANGE-PT=1" or set an escape route in the TPC display for the inside turning.



• TOOL

Select a tool to be used. See the relevant items in "Bar-Materials Machining Unit (BAR)" for tool selection method. For manual program machining unit, R-TOOL/F-TOOL distinction is not drawn. The NC regards the tool as R-tool.
If a tool is not designated here, the currently valid tool will be used as it is.

3-11-2 Setting sequence data

UNo.	UNIT	CHANGE-PT					GEAR	TOOL	
*	MNP	***					***	***	
					RADIUS/VARIABL				
SEQ	G	DATA-1	DATA-2	DATA-3	E	RPM	FEED	M	OFS
1	•	•	•	•	•	•	•	•	•

• G

The following menu will be displayed when the cursor is placed at this item.

G00	G01	G02	G03	G04	G32	G34		SHAPE END	
(a)	(b)	(c)	(d)	(e)	(f)	(g)			

From (a) through (g) above, select G-code to be used. The data of the displayed menu denote the following function.

G00 : Rapid feed positioning
G01 : Linear interpolation
G02 : Clockwise arc interpolation
G03 : Counterclockwise arc interpolation
G04 : Dwell
G32 : Fixed-pitch threading
G34 : Variable-pitch threading

Note: The codes G00, G01, G02, G03, G32 and G34 are modal information • information that remains valid until you have set any other G-code. The code G04 is unmodal information • information that becomes valid only for the sequence data you set.

• DATA-1, • DATA-2, • DATA-3

The following menu will be displayed when the cursor is placed at either one of these three items:

X	Z								

- If the G-code you have selected for item • above is other than G04:
  Select an axis from the menu shown above, and then specify the position to which the movement is to be performed on the axis.
- If the G-code you have selected for item • above is G04:
  After pressing the **X** menu key, set the desired dwell time in seconds.

For the manual program machining unit, use the following coordinate system to specify the position to which the cutting edge of the tool is to be moved.

For absolute programming, set the following data:

```
UNo.  UNIT  CHANGE-PT
 *    MNP       1
SEQ    G     DATA-1      DATA-2    DATA-3
 1    00     X  70.   Z    5.
 2    01     X  70.   Z  -35.
 3    02     X  70.   Z  -70.
 4    01     X  80.   Z  -95.
 5    01     X  80.   Z -105.
 6    03     X  80.   Z -135.
 7    01     X  80.   Z -145.
 8    01     X 130.   Z -145.
```

(*1) → SEQ 5
(*1) → SEQ 8

For incremental programming, set the following data:

```
UNo.  UNIT  CHANGE-PT
 *    MNP       1
SEQ    G     DATA-1      DATA-2    DATA-3
 1    00     X  70.      Z    5.
 2    01     X   0. <    Z  -40. <
 3    02     X   0. <    Z  -35. <
 4    01     X  10. <    Z  -25. <
 5    01     X   0. <    Z  -10. <
 6    03     X   0. <    Z  -30. <
 7    01     X   0. <    Z  -10. <
 8    01     X  50. <    Z   -0. <
```

(*1) → SEQ 5
(*1) → SEQ 8

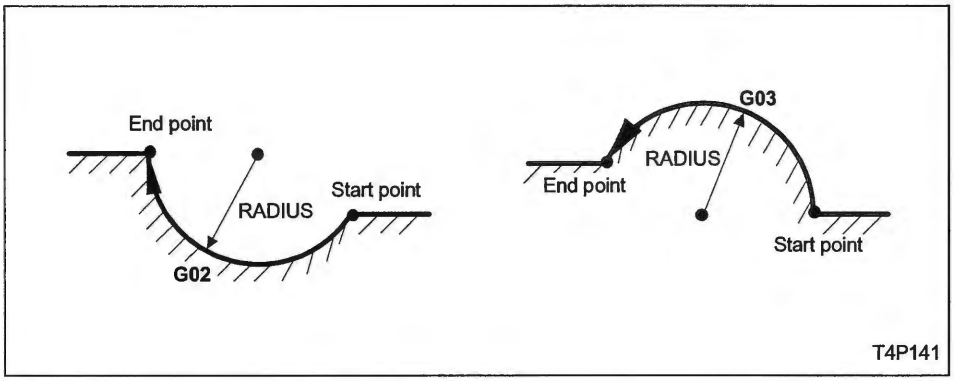
Incremental programming (*2)

*1: Since G01 is modal information, corresponding data can be omitted.

*2: Setting data using the incremental programming method displays the data with a < marking on the right side.

• RADIUS/VARIABLE

- If you have selected whether G02 or G03 for item • above:
  Set a radius for the desired arc.



- If you have selected G34 for item • above:
  Set the desired increment or decrement in thread pitch per revolution.
- If you have selected other G-code: No data can be set. (A • mark will be displayed.)

•RPM

Set a rotational speed (rpm) or peripheral speed for the spindle.

1) To set a rotational speed:
   Set the desired rpm value directly. An **S** marking will then be displayed to the left of that value.

```
RADIUS/VARIABLE   RPM      FEED
      □Y          S2000
                  □a
                Setting data
```

2) To set a peripheral speed:
   After reversing **SURF SPD V** display by pressing its menu key, set the desired value in m/min or ft/min units.
   A **V** marking will then be displayed to the left of that value.

```
RADIUS/VARIABLE   RPM      FEED
      □Y          V130
                  □a
                Setting data
```

3-12 M-Code Unit (M)

Select the M-code unit when M-codes (auxiliary function codes) are to be set.
Up to a maximum of 12 M-codes can be set for one M-code unit.
Press **M CODE** menu key for this unit.

3-12-1 Setting unit data (M-code)

UNo.	UNIT	#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12
*	M	•	•	•	•	•	•	•	•	•	•	•	•

• #1 to #12

Set the desired M-code number in each of the 12 items.
The M-codes you have set are executed in the following order:

#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12
***	***	***	***	***	***	***	***	***	***	***	***

1. (Synchronous) → 2. (Synchronous) → 3. (Synchronous)

If not all of the intended M-codes are to be executed at the same time, therefore, divide them into three groups (#1 through #4, #5 through #8, and #9 through #12) and then set those M-codes separately.

Note: For the list of M-codes provided in the NC system, see the M-code list given below.
Of all listed M-codes, those which can be actually selected differ according to the particular specifications of your machine.
The M-codes M02 (Program End), M98 (Subprogram Call), or other dedicated M-codes for EIA/ISO program cannot be selected.
All M-codes, including those for software options, are listed in the M-code list.

*: Option

M-code	Function	Remarks
M44	Spindle gear select High (For four-gearage)	
M45	Air coolant	*
M46	Turret one-way rotation	*
M47	Turret one-way rotation cancel	*
M48	Parts catcher forward	*
M49	Parts catcher backward	*
M50		
M51	Error detection invalid	
M52	Error detection valid	
M53	Chamfering off	
M54	Chamfering on	
M55	Piece count	*
M56	Front door open	*
M57	Front door close	*
M58	Air blast	*
M59	Program end	*
M60		
M61		
M62		
M63		
M64		
M65		
M66		
M67		
M68	Bar feeder command 1	*
M69	Bar feeder command 2	*
M70		
M71		
M72	Chuck internal clamping	*
M73	Chuck external clamping	*
M74	Workpiece rest unconnect	*
M75	Workpiece rest connect	*
M76	Chuck jaw unload (AJC)	*
M77	Chuck jaw load (AJC)	*
M78	Workpiece rest dog valid	*
M79	Workpiece rest dog invalid	*
M80		
M81	Workpiece measurement ON	*
M82	Workpiece measurement OFF	*
M83	Tool tip measurement ON	*
M84	Tool tip measurement OFF	*
M85		
M86	Workpiece rest-1 unclamp	*
M87	Workpiece rest-1 clamp	*
M88	Workpiece rest-2 unclamp	*
M89	Workpiece rest-2 clamp	*

*: Option

M-code	Function	Remarks
M136		
M137		
M138		
M139		
M140		
M141		
M142		
M143		
M144		
M145		
M146		
M147		
M148		
M149		
M150		
M151		
M152		
M153		
M154		
M155		
M156		
M157		
M158		
M159		
M160		
M161		
M162		
M163		
M164		
M165		
M166		
M167		
M168		
M169		
M170		
M171		
M172		
M173		
M174		
M175		
M176		
M177		
M178		
M179		
M180		
M181	Chuck jaw #1 select	*

*: Option

M-code	Function	Remarks
M226		
M227		
M228		
M229		
M230		
M231		
M232		
M233		
M234		
M235		
M236		
M237		
M238		
M239		
M240		
M241		
M242		
M243		
M244		
M245		
M246		
M247		
M248	Infeed start setup (Spindle speed monitoring)	*
M249	Tool change setup (Turret unclamping)	*
M250		
M251		
M252		
M253		
M254		
M255		
M256		
M257		
M258		
M259		
M260		
M261		
M262		
M263		
M264		
M265		
M266		
M267		
M268		
M269		
M270		
M271		

3-13 Subprogram Unit (SUB)

Select the subprogram unit to call up an EIA/ISO or a macro program as a subprogram from the MAZATROL program.
This unit is only available with the optional function for EIA/ISO or macro programming.
Press the **SUB PROGRAM** menu key to select this unit.

3-13-1 Setting unit data

```
UNo.   UNIT       WK.No.     NUM.
*      SUB        •          •
```

•WK. No.

Enter the work number of the desired subprogram.

• NUM.

Enter the number of repetitions of the subprogram.
If no data is entered here, the subprogram will be executed one time.

3-13-2 Setting sequence data

```
UNo.   UNIT       WK.No.     NUM.
*      SUB        ***        ***
SEQ    ARGM 1   ARGM 2   ARGM 3   ARGM 4   ARGM 5   ARGM 6
1      • •      • •      • •      • •      • •      • •
```

• • ARGM

First enter the address • and then input the data • of an argument in pairs.

- To set a macro variable as the argument data, press the **MACRO INPUT** menu key before entering the number of the macro variable. In this case, a symbol "#" is displayed before the numerical data.
- If no argument is required, press the **SEQUENCE END** menu key with the cursor placed at the position • under ARGM 1.

Note 1: The subprogram command mode comes into effect the moment that the tool has reached the safety profile position after machining of the preceding unit. The modal status at the beginning of the called subprogram is the same as the status after resetting.
All subprograms must terminate with an M99 command (return from subprogram).

Note 2: In the control mode where the coordinate system of the main program is conveyed to the subprogram (bit 5 of **P108** is set to 1), the Z-offset data registered in the **SET UP** display for the main program will remain valid for a subprogam in which the MAZATROL coordinate system is selected.

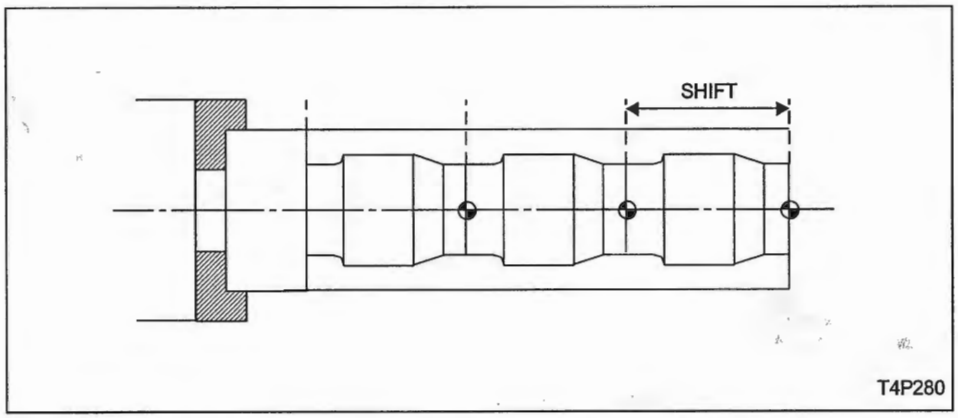
Note 3: Modification of coordinate system or modal information made in an EIA/ISO subprogram cannot simply be transferred to the MAZATROL main program.

• NUM.

If the current program is to be executed repeatedly, set the desired number of times of execution.

• SHIFT

Shifting the origin of the current program and repeatedly executing it enable multiple parts of the same shape, or a single part of identical recurring shape patterns as shown in diagram below, to be made from one workpiece.
For such machining, set the desired shift amount of the program origin in this item.



Note 1: Data set in this item becomes valid only if the number of times of program execution has been set in item • above.

Note 2: If no data has been set in this item, the shift amount will be regarded as 0.

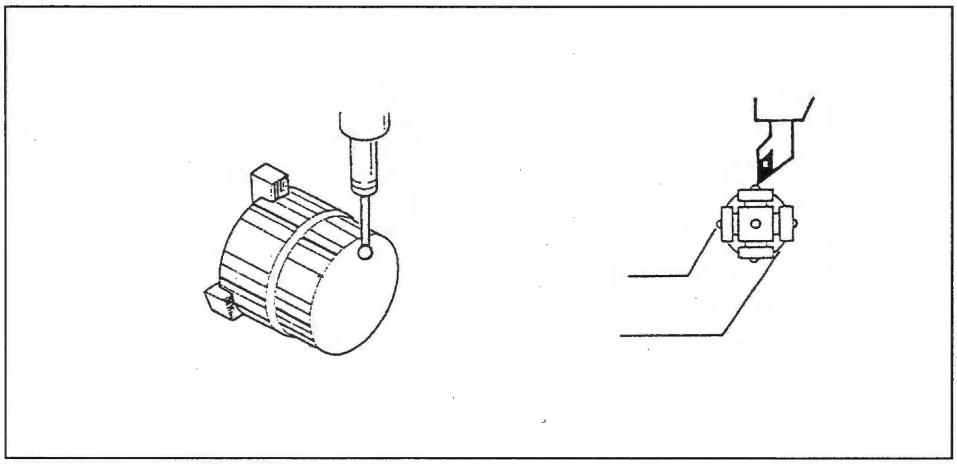
Note 3: Data must not be set in this item if a measurement unit (MES) is to be executed. Setting data other than 0 may cause contact between the touch sensor and the workpiece.

Note 4: As for repetitive machining on a single workpiece, the following condition must be satisfied:

LENGTH (Common data) > NUM. × SHIFT (END unit) + WORK FACE (Common data)

3-15 Measurement Unit (MES)

Select the measurement unit to measure the workpiece dimensions using a touch sensor or the wear amount of a tool using the TOOL EYE. (The measurement function is an option.)



Press the **MEASURE** menu key to select this unit.

DIA BOTH SIDES	STEP	GROOVE	WIDTH	DISTANCE ONE SIDE	TOOL	EXTERNAL	Z-AXIS OFFSET		

Select a measurement type from the menu shown above.

DIA BOTH SIDES : To measure the diameter of machined workpiece.

STEP : To measure the step distance of machined workpiece.

GROOVE : To measure the width of groove or other recesses.

WIDTH : To measure the width of protrusion or other convexities.

DISTANCE ONE SIDE : To measure the relative position of a point in relation to the preset reference point. Useful for measuring the outside diameter of a center workpiece machined with tailstock.

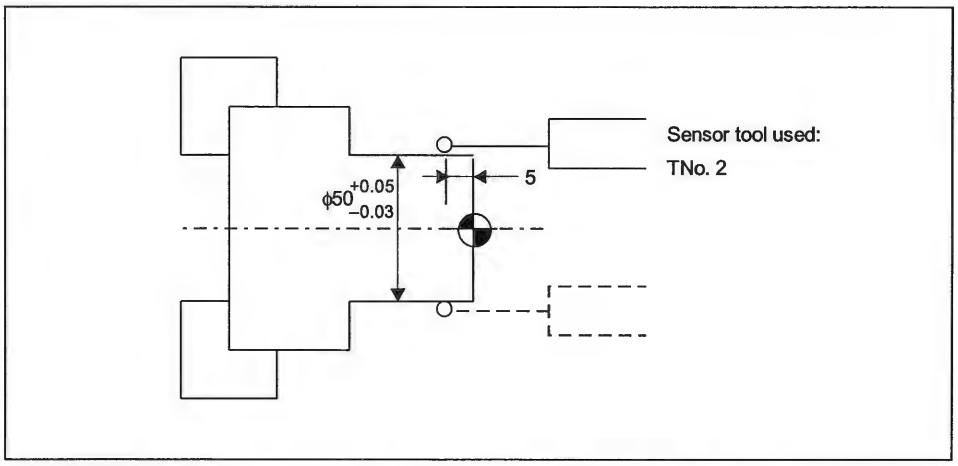
TOOL : To measure the tool nose.

EXTERNAL : To read the measurement data of an external measuring unit. (During this unit, the data is only read in for compensating the tool data registered in the NC unit; no machine action takes place.)

Z-AXIS OFFSET : To measure the position of workpiece face for setting correct Z-offset data.

Eight types of measurement are provided for the measurement unit. Refer to the following descriptions for the unit and sequence data to be set for execution of each measurement type.

C. Example program of MES DIA unit

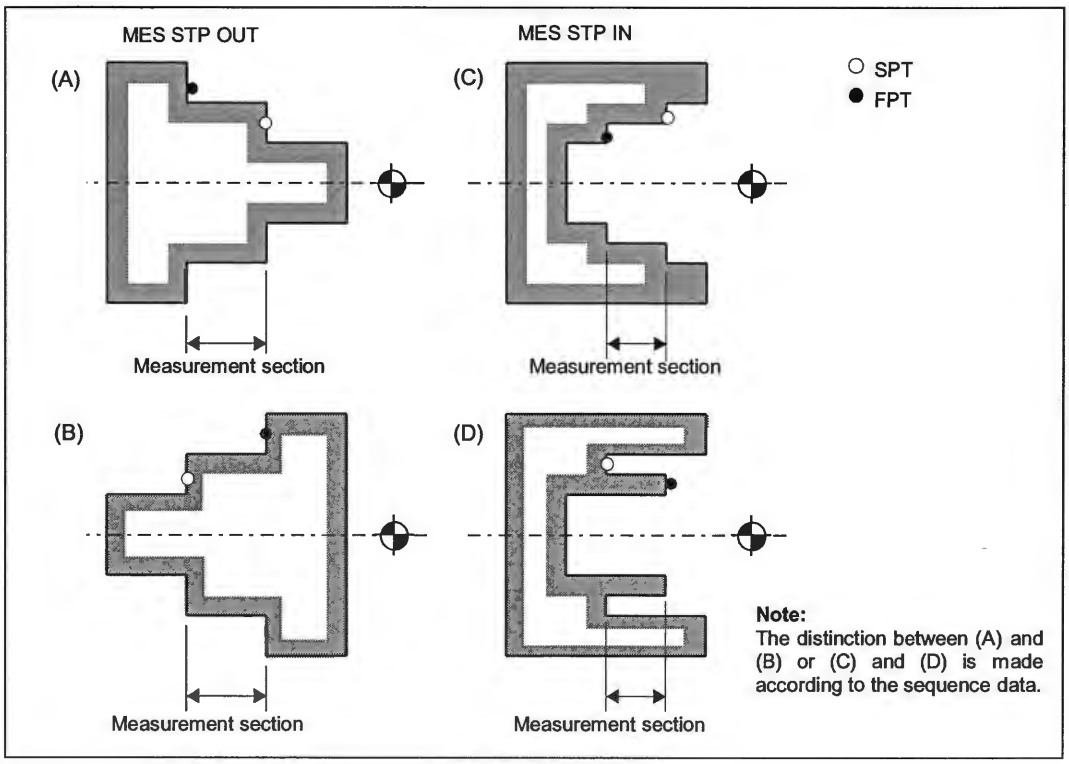


UNo.	UNIT	PART		OFS-TOOL	SNS-TOOL
*	MES DIA	OUT		1	2
SEQ	SPT-X	SPT-Z	T LIM+	T LIM-	BASE
1	50.	5.	0.05	-0.03	0

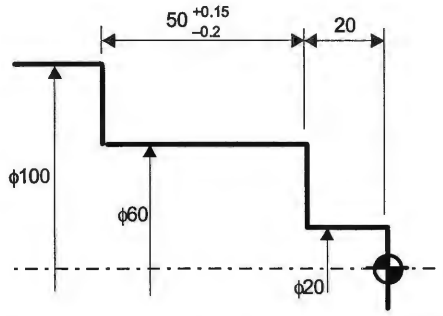
2. STP (Step distance measurement; STEP)

A. Unit data

UNo.	UNIT	PART	OFS-TOOL	SNS-TOOL
*	MES STP	•	•	•

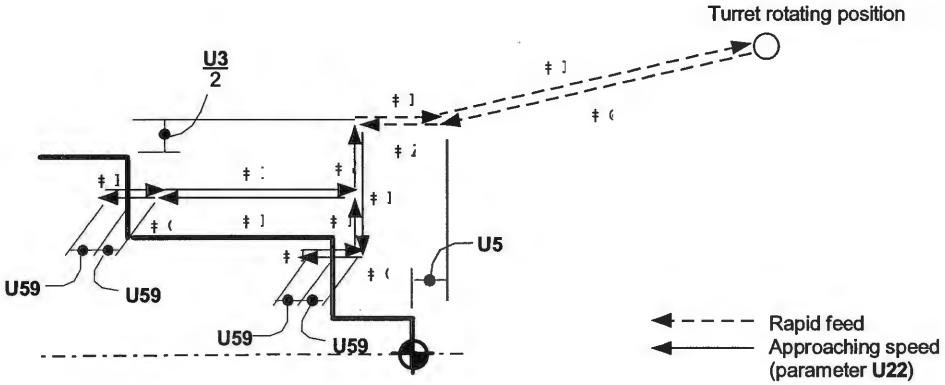


C. Example program and tool path of MES STP unit



Unit: mm

UNo.	UNIT	PART				OFS-TOOL	SNS-TOOL
*	MES STP	OUT				2	1
SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	T LIM+	T LIM-	BASE
1	50.	20.	80.	70.	0.15	-0.2	0

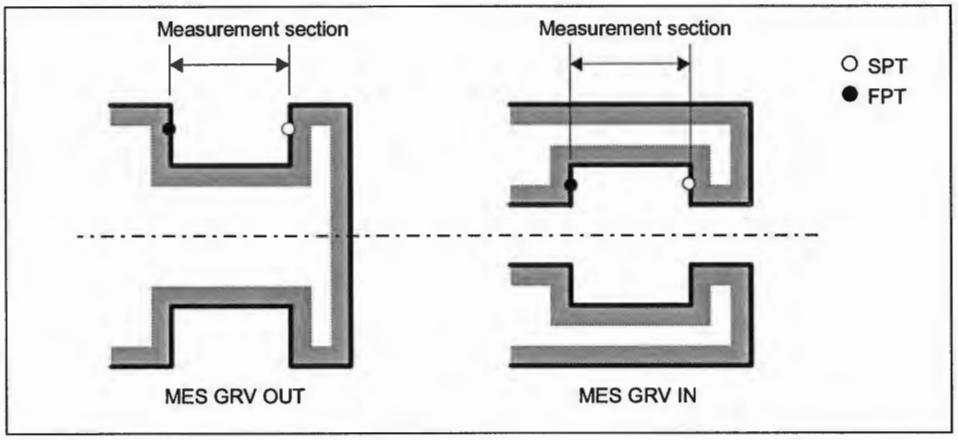


U3 : Safety profile clearance (O. D.)
U5 : Safety profile clearance (Face)
U59 : Workpiece measurement stroke;
Movement through the stroke at measuring speed (**U21**)

3. GRV (Groove width measurement; GROOVE)

A. Unit data

UNo.	UNIT	PART	OFS-TOOL	SNS-TOOL
*	MES GRV	•	•	•



• PART

Select the item **OUT OUTER DIAMETER** or **IN INNER DIAMETER** from the menu (see the figure above).

• OFS-TOOL, • SNS-TOOL

Refer to the description for **MES DIA**.

B. Sequence data

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	T LIM+	T LIM-	BASE
1	(a)	(b)	(c)	(d)	(e)	(f)	(g)

(a) SPT-X, (b) SPT-Z

Specify the X- and the Z-coordinates of the starting measurement point.

(c) FPT-X, (d) FPT-Z

Specify the X- and the Z-coordinates of the ending measurement point.

(e) T LIM+, (f) T LIM–

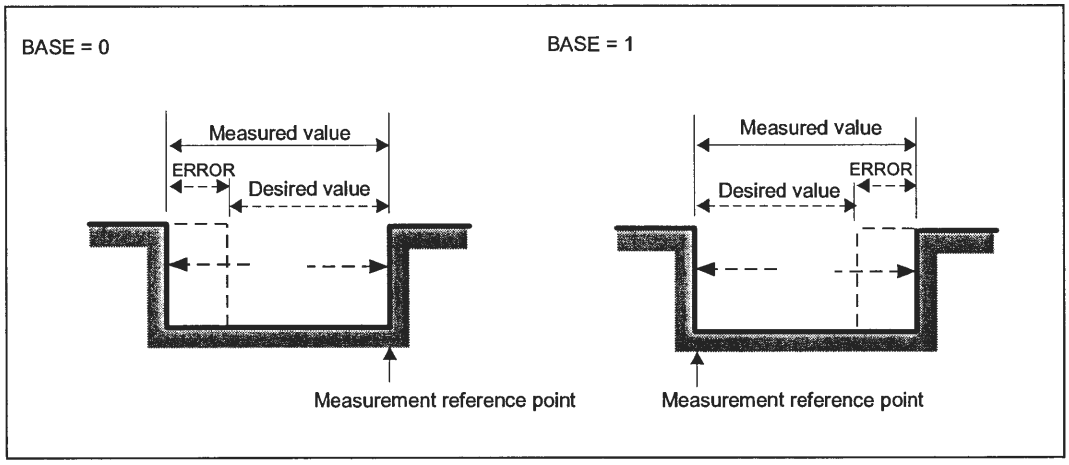
Specify the upper and the lower tolerance limit.

(g) BASE

Specify the measurement reference point.
Enter "0" if the one measurement point lying on the right of the other point is to be set as reference, and specify "1" for the other (refer to subsection D).

D. Compensation

An example in which the measured value of GRV OUT is larger than the desired value.



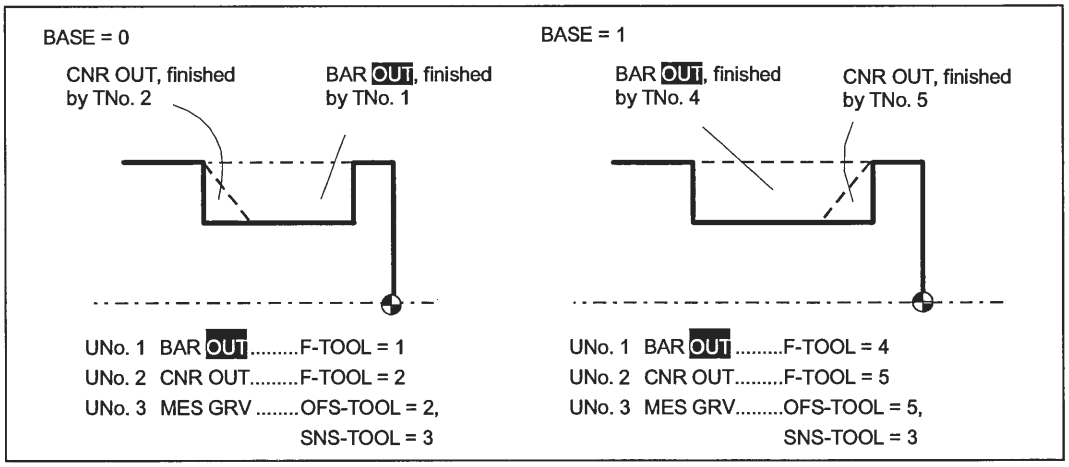
In the example shown above:

- For BASE = 0, compensation is made to the right since the workpiece has been excessively cut to the left (seen from the reference point).
- For BASE = 1, compensation is made to the left since the workpiece has been excessively cut to the right.

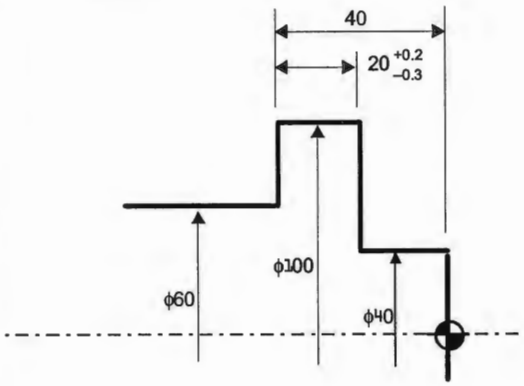
Note: Conversely, if the measured value is smaller than the desired value:
- For BASE = 0, compensated to the left,
- For BASE = 1, compensated to the right.

<Supplement>

Example of program construction and setting for BASE.
The MES GRV unit is really effective only for a measurement in which the positions of two surfaces machined by various tools are to be measured. The compensation for one and the same tool by which the both surfaces are to be machined will only shift the machining shape as it is.

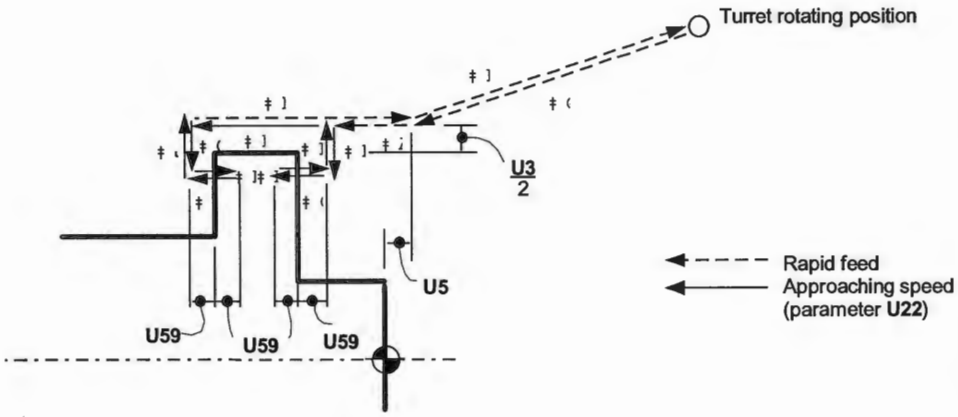

BASE = 0
CNR OUT, finished by TNo. 2
BAR OUT, finished by TNo. 1
UNo. 1 BAR OUT.....F-TOOL = 1
UNo. 2 CNR OUT.....F-TOOL = 2
UNo. 3 MES GRV .....OFS-TOOL = 2,
SNS-TOOL = 3
BASE = 1
BAR OUT, finished by TNo. 4
CNR OUT, finished by TNo. 5
UNo. 1 BAR OUT .....F-TOOL = 4
UNo. 2 CNR OUT.....F-TOOL = 5
UNo. 3 MES GRV.....OFS-TOOL = 5,
SNS-TOOL = 3

C. Example program and tool path of MES WID unit



Unit: mm

UNo.	UNIT	PART				OFS-TOOL	SNS-TOOL
*	MES WID	OUT				2	1
SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	T LMT+	T LMT-	BASE
1	90.	20.	90.	40.	0.2	-0.3	0

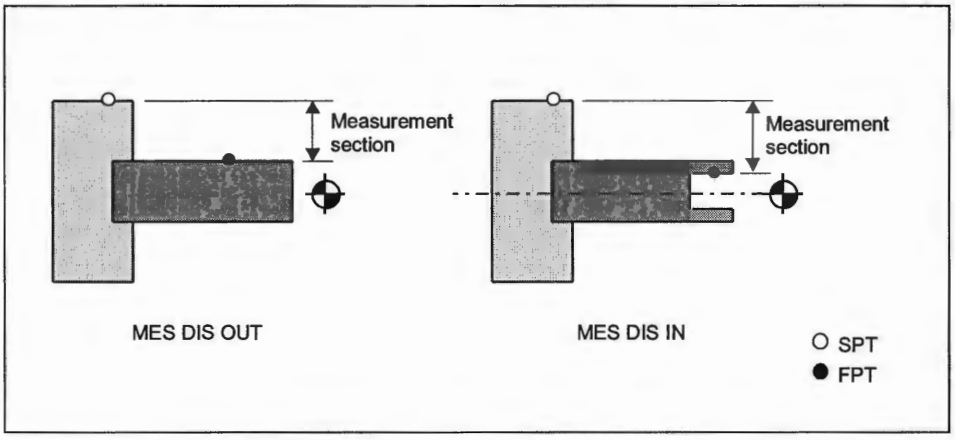


U3 : Safety profile clearance (O. D.)
U5 : Safety profile clearance (Face)
U59 : Workpiece measurement stroke;
Movement through the stroke at measuring speed (**U21**)

5. DIS (Distance measurement; DISTANCE ONE SIDE)

A. Unit data

UNo.	UNIT	PART	OFS-TOOL	SNS-TOOL
*	MES DIS	•	•	•



• PART

Select the **OUT OUTER DIAMETER** or **IN INNER DIAMETER** from the menu (see the figure above).

• OFS-TOOL, • SNS-TOOL

Refer to the description for MES DIA.

B. Sequence data

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	T LIM+	T LIM-	BASE
1	(a)	(b)	(c)	(d)	(e)	(f)	(g)

(a) SPT-X, (b) SPT-Z

Specify the X- and the Z-coordinates of the starting measurement point.

(c) FPT-X, (d) FPT-Z

Specify the X- and the Z-coordinates of the ending measurement point.

(e) T LIM+, (f) T LIM–

Specify the upper and the lower tolerance limit.

(g) BASE

Always enter "0" for the MES DIS unit.

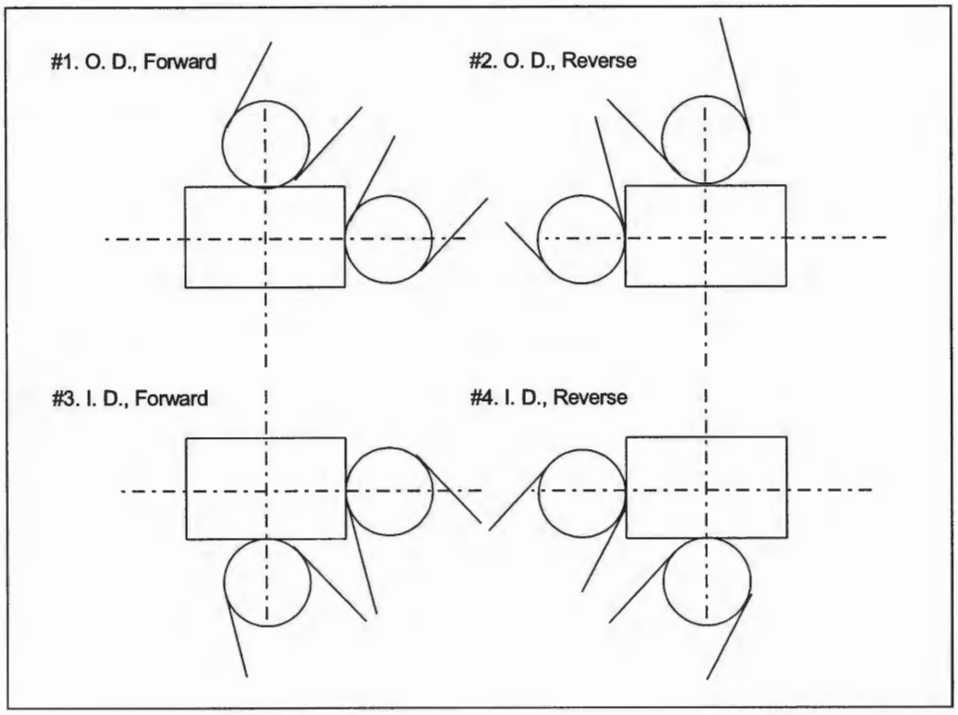
6. TOL (Tool-nose measurement; TOOL)

A. Unit data

UNo.	UNIT	PART	OFS-TOOL	SNS-TOOL
*	MES	•	•	•
	TOL			

• PART

Specify the type of the tool to be measured with 1 to 4.



• OFS-TOOL

Enter the identification number of the tool to be measured.

B. Sequence data

SEQ	T-LIM-X	T-LIM-Z	TOOL EYE
1	(a)	(b)	(c)

(a) T-LIM-X, (b) T-LIM-Z

Specify the maximum amount of compensation in the X- and Z-axis direction.

- The measurement will only be executed for the axis for which the T LIM data is specified here.

(c) TOOL EYE

Enter "0" if the tool-nose measuring unit is to be returned at the end of the measurement unit, or set "1" to leave the unit in the measurement position.

- If another tool is to be measured in succession, enter "1" to save the operation time. Always set "0", however, for the last MES TOL unit not to terminate the unit with the measuring unit remaining in a position improper for the next unit.

therefore, as data for "memory (coment)".

8. ZOF (Z-offset measurement; Z-AXIS OFFSET)

A. Unit data

UNo.	UNIT	PART	OFS-TOOL	SNS-TOOL
*	MES ZOF	•	•	•

• SNS-TOOL

Enter the tool number of the touch sensor.

B. Sequence data

SEQ	SPT-X	SPT-Z	TARGET-Z	SHIFT-Z	RETURN	MOVE
1	(a)	(b)	(c)	(d)	(e)	(f)

(a) SPT-X, (b) SPT-Z

Specify the X- and the Z-coordinates of the measurement starting point.
The starting point refers to the approaching position for the proper MES ZOF unit.
According to the respective positional relation between tool and workpiece, this approach will be performed through a relay point which accords with the safety profile clearance.

(c) TARGET-Z

Specify the Z-coordinate (desired value) of the measurement point (sensor-ON position).

(d) SHIFT-Z

Specify the Z-axis value which shall denote the position of the measured point in the new offset (compensated) system of coordinates (Normally, the value of SHIFT-Z should be equal to TARGET-Z.).

(e) RETURN

Specify the stroke through which the touch sensor is to be returned from the skipping position of temporary measurement at approaching speed.
If "0" is set here, only temporary measurement at the measurement approaching speed will be executed (not followed by the measurement at measuring speed).

(f) MOVE

Specify the maximum movement distance of the touch sensor over the desired position.

Rapid return on Z-axis to position SPT-Z.

- A•
  Simultaneous rapid movement on both axes to the tool-change position.

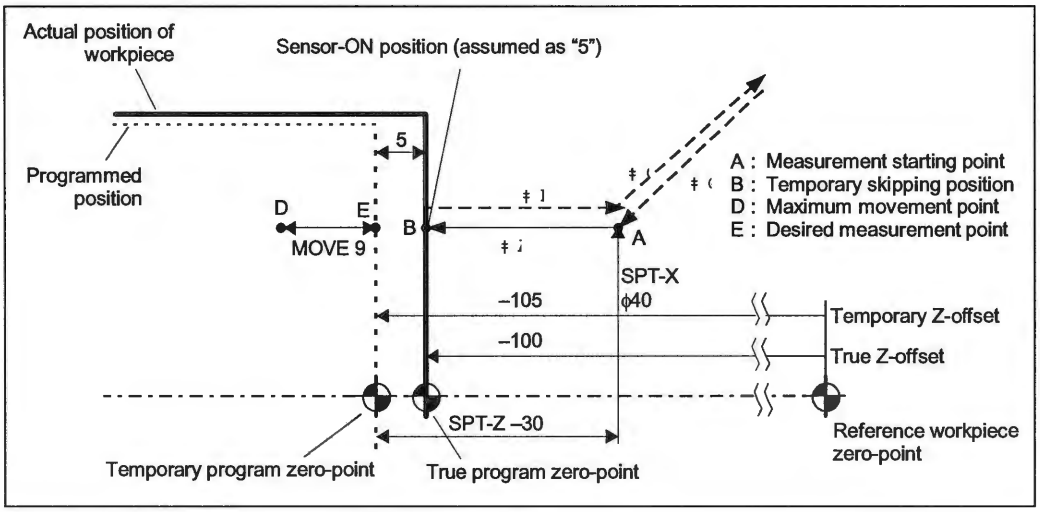
Note: If the skip signal does not turn on even upon arrival at the maximum movement point (D), a measurement alarm is displayed, which results in reset stop of the system.

- Compensation = – SHIFT-Z – Measured value (B') = 0 – 10= –10.

- True Z-offset = Temporary Z-offset – Compensation = –110 – (–10) = –110 + 10 = –100.

Example 2: TARGET-Z = 0 (no face protrusion), RETURN = 0

UNo.	UNIT	PART			OFS-TOOL	SNS-TOOL
*	MES ZOF	•			•	1
SEQ	SPT-X	SPT-Z	TARGET-Z	SHIFT-Z	RETURN	MOVE
1	40.	-30.	0.	0.	0.	9.



Operation

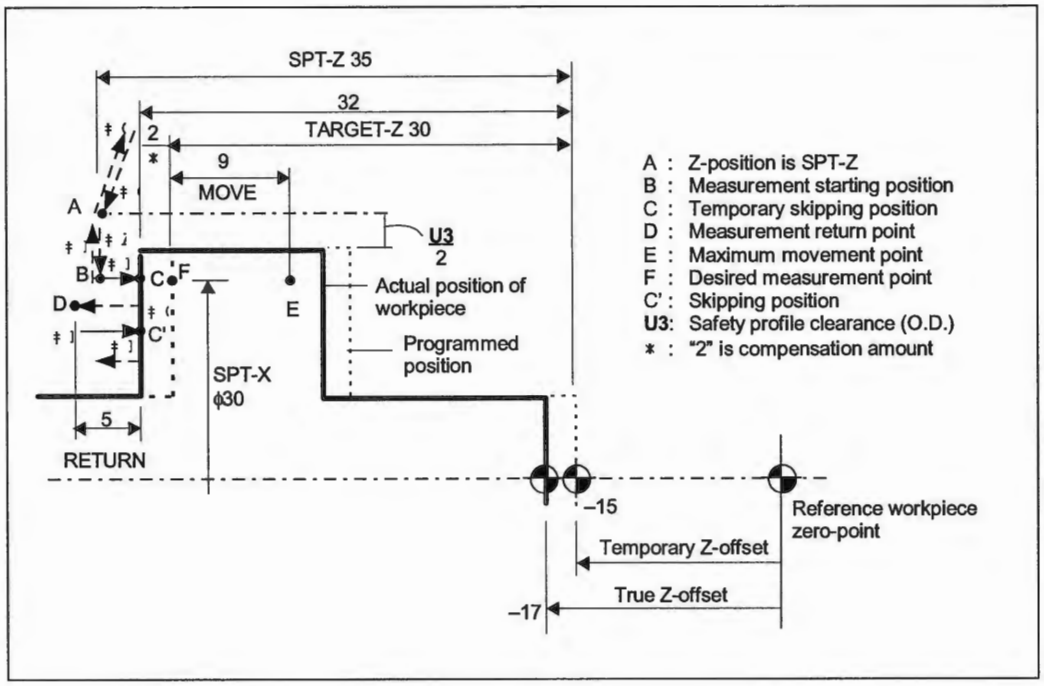
- •A
  Simultaneous rapid movement on X- and Z-axis to the measurement starting point.
- A•B
  Measurement at the speed of parameter **U22**.
  The data of skipping position B is used for calculation of Z-offset value.
- B•A
  Rapid return on Z-axis to position SPT-Z.
- A•
  Simultaneous rapid movement on both axes to the tool-change position.

- Compensation = – SHIFT-Z – Measured value = 0 – 5 = –5.

- True Z-offset = Temporary Z-offset – Compensation = –105 – (–5) = –100.

Example 4: TARGET-Z = 30, SHIFT-Z = 30, RETURN = –5
(Reverse direction measurement; SPT-Z > TARGET-Z)

UNo.	UNIT	PART			OFS-TOOL	SNS-TOOL
*	MES ZOF	•			•	1
SEQ	SPT-X	SPT-Z	TARGET-Z	SHIFT-Z	RETURN	MOVE
1	30.	35.	30.	30.	-5.	9.



Operation

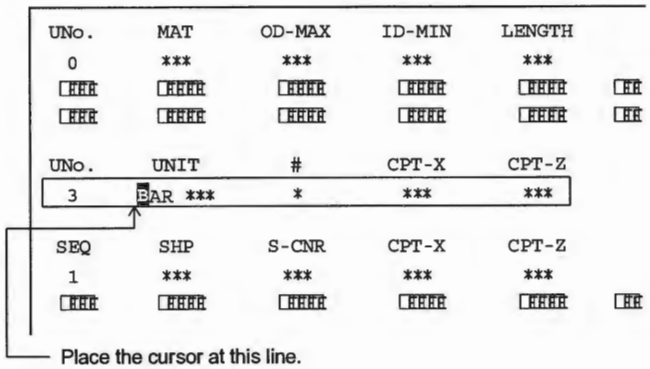
- •A
  Simultaneous rapid movement on X- and Z-axis to the point of (O.D. clearance, SPT-Z). This relay point is set to avoid the collision with the workpiece.
- A•B
  Rapid movement on X-axis to position SPT-X.
- B•C (B•E)
  Movement on Z-axis towards TARGET-Z at the speed of parameter **U22** (Temporary measurement). Maximum movement to point E (= TARGET-Z + MOVE).
- C•D
  Rapid return from the skipping position C through RETURN distance.
- D•C'
  Measurement at the speed of parameter **U21**.
  The data of skipping position C' is used for calculation of Z-offset value.
- C'•B
  Rapid return on Z-axis to position SPT-Z.
- B•A
  Rapid return on X-axis to O.D. clearance.

4 TPC DATA SETTING

4-1 Operating Procedure for Setting TPC (Tool-Path Control) Data

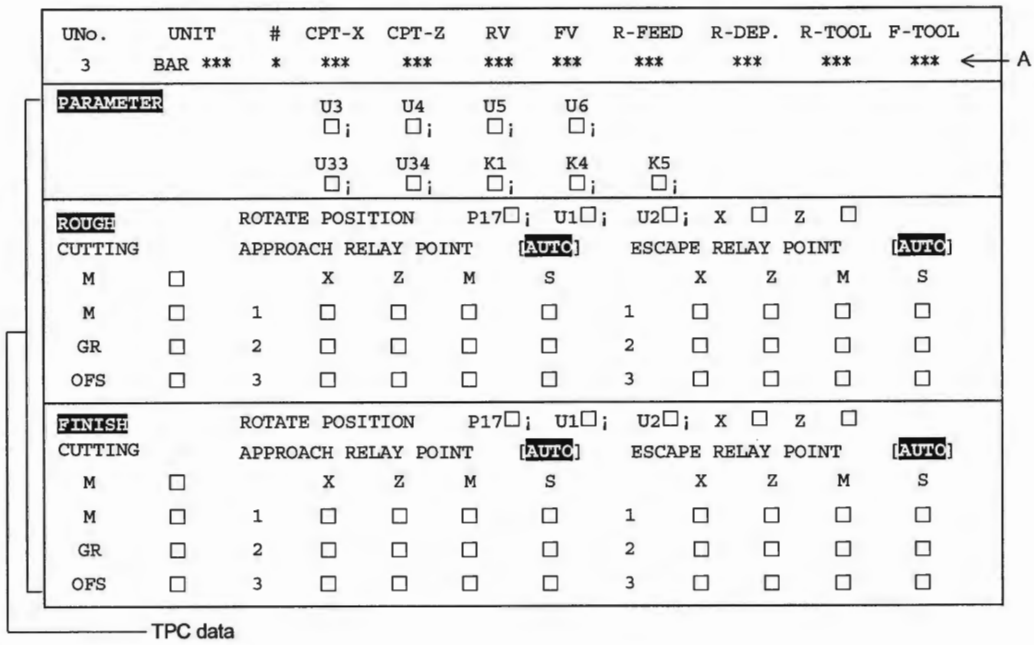
(1) First, place the cursor at the unit data line of the unit for which the TPC data is to be set.

Example: To set TPC data for the bar-materials machining unit (BAR) (UNo. 3):



(2) Press the menu selector key, and then the **TPC** menu key.

- Pressing the **TPC** menu key indicates the **TPC** display for the unit specified at step (1). For the case of the example above, the following **TPC** display will be presented on the screen.


UNo. UNIT # CPT-X CPT-Z RV FV R-FEED R-DEP. R-TOOL F-TOOL
3 BAR *** * *** *** *** *** *** *** *** ***
A
PARAMETER
U3 U4 U5 U6
U33 U34 K1 K4 K5
ROUGH CUTTING
ROTATE POSITION P17 U1 U2 X Z
APPROACH RELAY POINT [AUTO]
ESCAPE RELAY POINT [AUTO]
M X Z M S X Z M S
FINISH CUTTING
TPC data

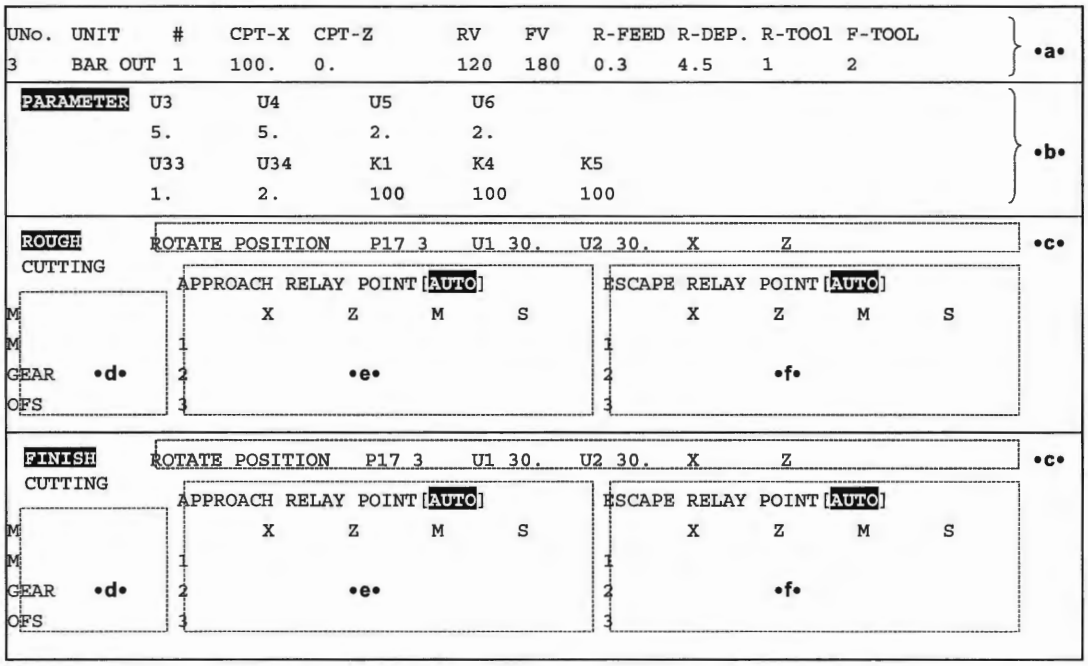
(3) Set the data in indicated item or change the data displayed in indicated item.

- On line (A) above, unit data that was set on the **PROGRAM** display will be displayed as it is. The data cannot be changed on the **TPC** display.
- Data that is preset in related parameters will be displayed at items marked with •. Parameters denoting the distance are usually preset in 0.001 mm (0.0001 inch) steps, but on the **TPC** display, they will be displayed in 1 mm (1 inch) steps.

In that case, you must carry out the procedure above (described in Note 3) to temporarily initialize the TPC data. Subsequently, you can set (or change) the desired TPC data once again. An alarm will occur if you make an attempt to execute the program with the ! mark displayed.

4-2 Description of Each TPC Data Item for Turning Units and Measurement Units

1. TPC data for turning units



(a) Unit data for which the **TPC** display is called up. The data cannot be changed with the TPC data displayed on the screen.

(b) The addresses of related parameters and the data that have been set on the **PARAMETER** display are displayed according to the particular type of unit. Modification of the data allows the machine to be correspondingly operated only during that unit. The data settings on the **PARAMETER** display, however, will not change by their modification on the **TPC** display. Refer to the separate Parameter List for details of parameter data.
Parameters denoting the distance are usually set in 0.001 mm (or 0.0001 in.) steps, but they will be displayed here in 1 mm (or 1 in.) steps.

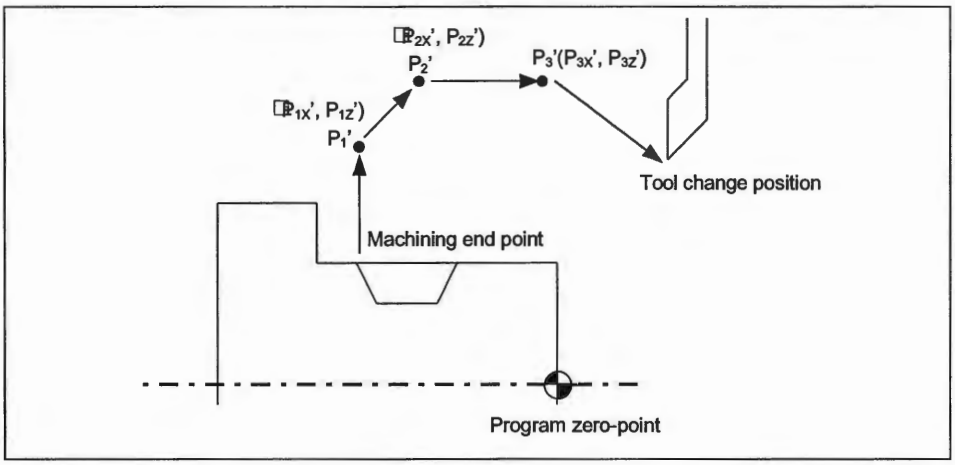
(c) Data related to the position of tool change (turret rotation) for each roughing or finishing unit.

- For items X and Z, specify the coordinates (in the machine coordinate system) of the required fixed point in mm (or in.).
- Refer to the Parameter List for details of P17, U1 (X) and U2 (Z).

(d) Data to be executed prior to the roughing or finishing unit.

- If two M-codes are set here, they will be output at the same time.
- Specify the required gear-No. in GR item. If no data is set here, a proper number will automatically be selected (which number will not be displayed here).
- For OFS item, enter an offset-No. of the data registered on the **TOOL OFFSET** display as required. For tool positioning, the data specified here will be taken into account in addition to TOOL SET and WEAR COMP. data on the **TOOL DATA** display.

(f) Use this section to modify the escape path for roughing or finishing so that interference does not occur.
Refer to the description in (e) for details on data setting.



ESCAPE RELAY POINT [MANU]

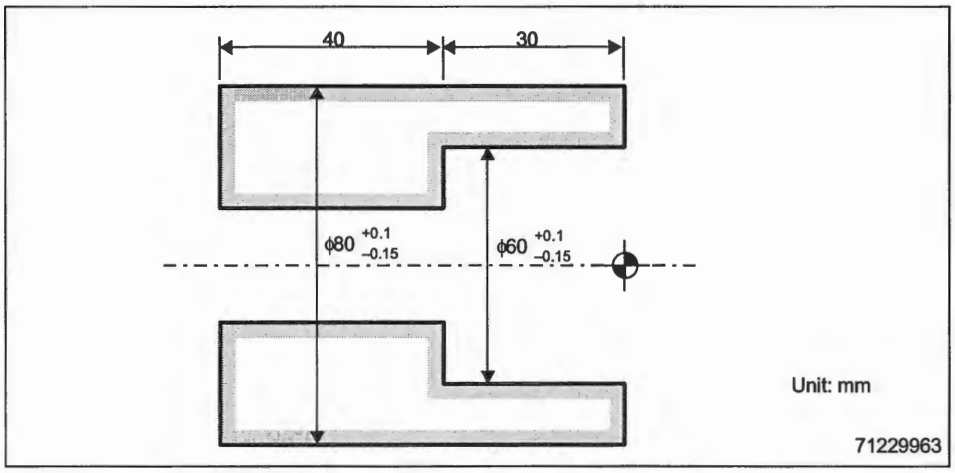
	X	Z	M	S	
1	P_{1X}'	P_{1Z}'			• Path through P_1'
2	P_{2X}'	P_{2Z}'	9		• Path through P_2', coolant OFF
3	P_{3X}'	P_{3Z}'		100	• Path through P_3', spindle speed 100 rpm

Note 1: In MES TOL (tool edge measurement) unit, the tool change position is determined by the parameter **A6**. Therefore, the tool change position (ROTATE POSITION) given on the **TPC** display will be ignored.

Note 2: The manually input data for relay points are not cancelled by changing [MANU] over to [AUTO]. They will automatically be restored by changing [AUTO] back to [MANU] again. To change the data, therefore, first clear the displayed data with the data cancellation key and then input new data as required.
Machine operation is always carried out according to the data setting on the **TPC** display.

2. TPC data for mesurement units

A. MES DIA program examples and tool paths



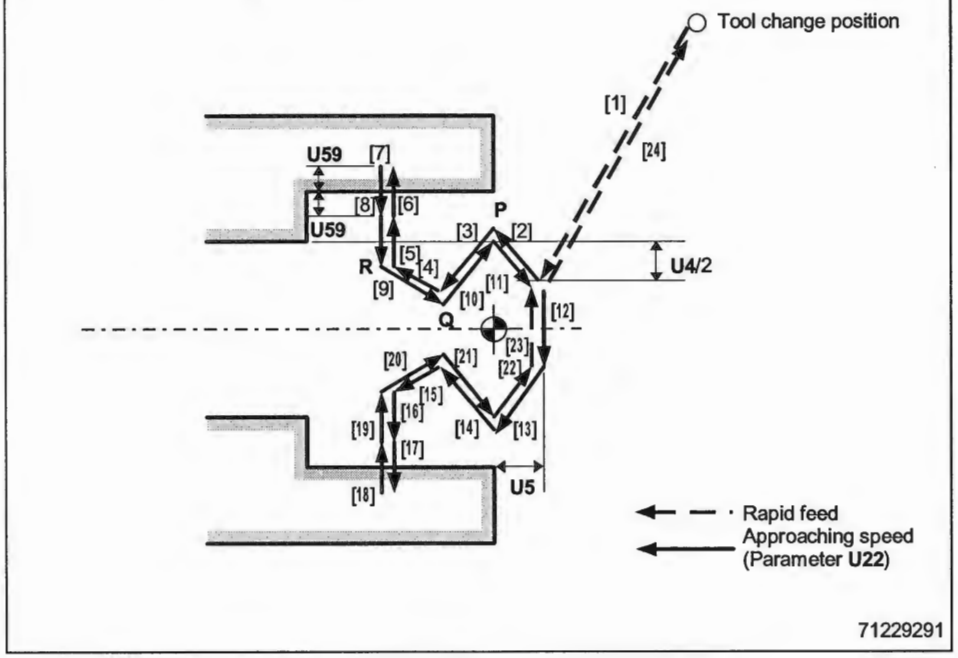
Example 2:

PROGRAM display

UNo.	UNIT	PART		OFS-TOOL	SNS-TOOL
**	MES DIA	IN		3	1
SEQ	SPT-X	SPT-Z	T LIM+	T LIM-	BASE
1	60	20	0.1	-0.15	0

TPC display

RELAY	X	Z
POINT 1	50	0
2	20	10
3	30	20



U4: Safety clearance from stock material (I. D.)

U5: Safety clearance from stock material (End stroke)

U59: Workpiece measurement stroke
When the tool moves through the workpiece measurement stroke that was set as parameter **U59**, the tool assumes the measuring speed that was set as parameter **U21**.

On the tool paths [6] and [17] the axis stops moving at the position where the touch sensor has turned on.

P, **Q**, **R**: Intermediate relay points that have been set on the **TPC** display

5 PROGRAM EDITING

This chapter describes operating procedures for editing programs already created. It also describes the various editing functions of the NC unit.

<Operating procedures for editing a MAZATROL program>

(1) Call up the **PROGRAM** display.
- Press the display selector key and then the **PROGRAM** menu key.

(2) Specify the work number of the program to be edited.
- After pressing the **WORK No.** menu key, specify the work number.
  The work-No. can also be specified in the work-Nos. listing window which will appear after pressing the above menu key. Using the cursor keys, place the cursor on the desired work-No. and press the input key.
- The selected program will be displayed on the screen.

(3) Press the **PROGRAM** menu key.
- Now, program data setting is possible; you can move the cursor to the desired position in the program using the cursor keys. Unless you press this menu key, you cannot change the program data being displayed; the cursor will only move vertically through the left end of each line even if you press the cursor keys.

(4) Edit the program.

(5) When necessary corrections or other edits to the program are completed, press the **PROGRAM COMPLETE** menu key.

<Program editing functions>

Pressing the menu changeover key with the **PROGRAM** display remaining on the screen displays the following menu:

PROGRAM COMPLETE	FIND	COPY	INSERT	ERASE		TPC	CALCULAT	SHAPE CHECK	HELP
	1	2	3	4					

You can use functions 1 to 4 above to carry out the editing operations listed below.

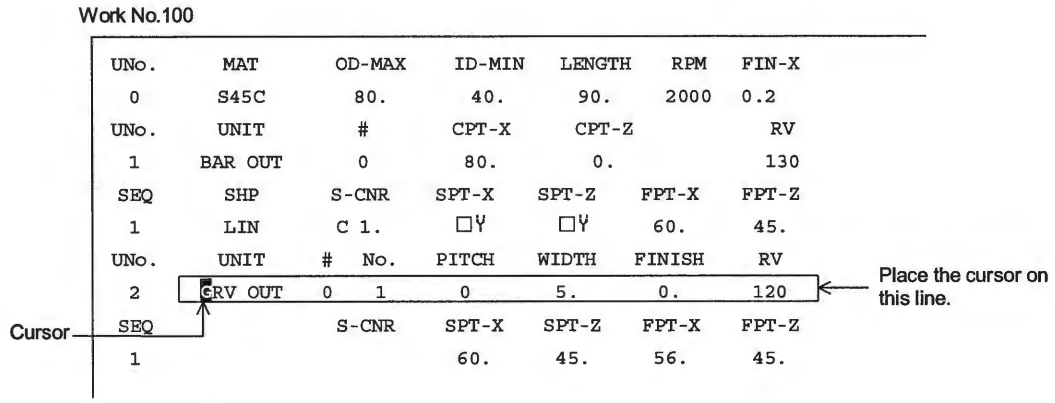
No.	Menu item	Function
1	FIND	To search for the intended unit in the program
2	COPY	To copy the data in unit of program, unit or machining shape (sequence)
3	INSERT	To insert one blank line for unit or sequence into any position on the **PROGRAM** display
4	ERASE	To delete a specific unit or sequence existing in the program currently being displayed on the **PROGRAM** display

5-2 Copy

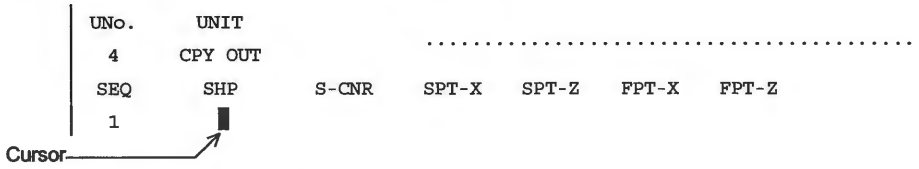
(1) Using the cursor keys, select the position where the copied data are to be placed.

- For program or unit copy, place the cursor on the line of the unit data which shall follow the copied data.
- For machining-shape copy, place the cursor at the first item of SEQ 1.

Example 1: To copy program or unit data into the position before unit No. 2:


Work No.100
UNo. MAT OD-MAX ID-MIN LENGTH RPM FIN-X
0 S45C 80. 40. 90. 2000 0.2
UNo. UNIT # CPT-X CPT-Z RV
1 BAR OUT 0 80. 0. 130
SEQ SHP S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 LIN C 1. 60. 45.
UNo. UNIT # No. PITCH WIDTH FINISH RV
2 CRV OUT 0 1 0 5. 0. 120
SEQ S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 60. 45. 56. 45.
Cursor
Place the cursor on this line.

Example 2: To copy machining-shape data into UNo. 4:



(2) Press the **COPY** menu key.

For subsequent steps, proceed to 5-2-1, 5-2-2 or 5-2-3 according to the type of data to be copied.

5-2-2 Unit copy

To copy only a specific unit existing in the program being displayed or in any other program pre-registered in the NC unit

(3) Press the **UNIT COPY** menu key.

(4) Set the work number of the program that includes the unit to be copied.

- Simply press the input key for the program being displayed.

(5) Set the number of the unit to be copied.

- The whole data (including sequence data) of the selected unit will be copied in the same position as for program copy (refer to 5-2-1).

Note: The common unit cannot be copied.

5-2-3 Machining-shape copy

To copy only a specific set of machining-shape data existing in the program being displayed

(3) Press the **SEQUENCE COPY** menu key.

(4) Set the number of the unit that includes the machining-shape data to be copied.

Example:

```
UNo.   UNIT      •••••••
                 •••
3      BAR OUT
SEQ    SHP       S-CNR     SPT-X  SPT-Z  FPT-X   FPT-Z
1      LIN       C 1.      •      •      50.     20.
2      TPR       C 1.      60.    20.    70.     40.
3      ▟                   70.    40.    80.     80.
UNo.   UNIT      •••••••
                 •••
4      CPY OUT
SEQ    SHP       S-CNR     SPT-X  SPT-Z  FPT-X   FPT-Z
1      ▌
```

If unit number 3 is set here:

```
UNo.   UNIT      •••••••
                 •••
3      BAR OUT
SEQ    SHP       S-CNR     SPT-X  SPT-Z  FPT-X   FPT-Z
1      LIN       C 1.      •      •      50.     20.
2      TPR       C 1.      60.    20.    70.     40.
3      ▟                   70.    40.    80.     80.
UNo.   UNIT      •••••••
                 •••
4      CPY OUT
SEQ    SHP       S-CNR     SPT-X  SPT-Z  FPT-X   FPT-Z
1      LIN       C 1.      •      •      50.     20.
2      TPR       C 1.      60.    20.    70.     40.
3      ▟                   70.    40.    80.     80.
```

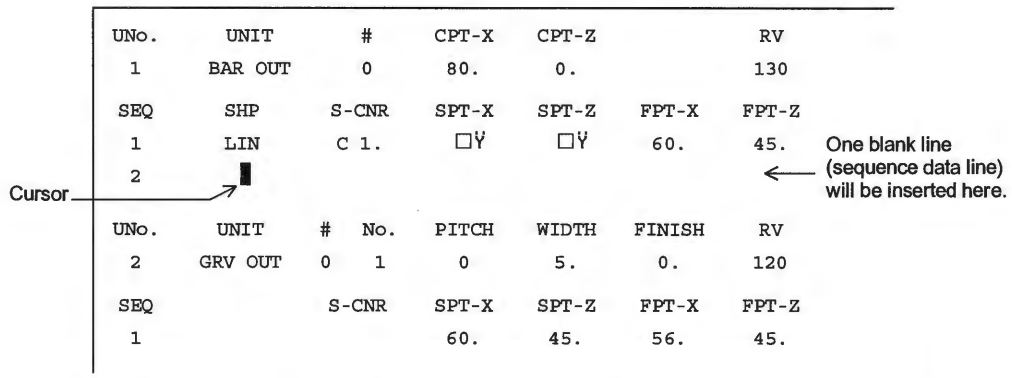
Machining-shape copy is available not only for an unit of the same type (MNP and MMP included) but also between the following units:

BAR ↔ CPY
DRL ↔ TAP

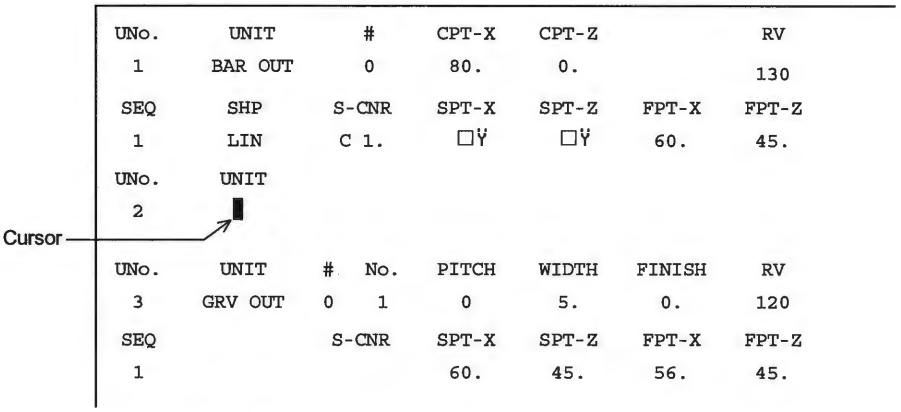
(3) Press the input key.

- One blank line will inserted.

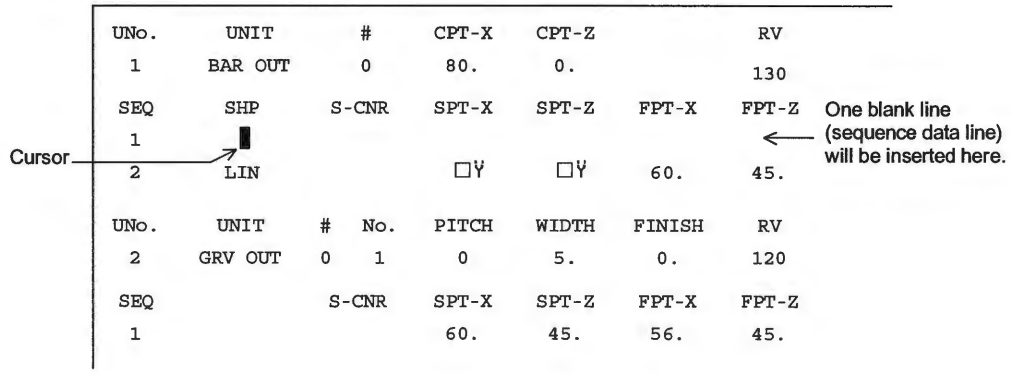
In case of example 1:


UNo. UNIT # CPT-X CPT-Z RV
1 BAR OUT 0 80. 0. 130
SEQ SHP S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 LIN C 1. □Ÿ □Ÿ 60. 45.
2
Cursor
One blank line (sequence data line) will be inserted here.
UNo. UNIT # No. PITCH WIDTH FINISH RV
2 GRV OUT 0 1 0 5. 0. 120
SEQ S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 60. 45. 56. 45.

Press the **SHAPE END** menu key, and the inserted line will be changed into an unit data line.


UNo. UNIT # CPT-X CPT-Z RV
1 BAR OUT 0 80. 0. 130
SEQ SHP S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 LIN C 1. □Ÿ □Ÿ 60. 45.
UNo. UNIT
2
Cursor
UNo. UNIT # No. PITCH WIDTH FINISH RV
3 GRV OUT 0 1 0 5. 0. 120
SEQ S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 60. 45. 56. 45.

In case of example 2:

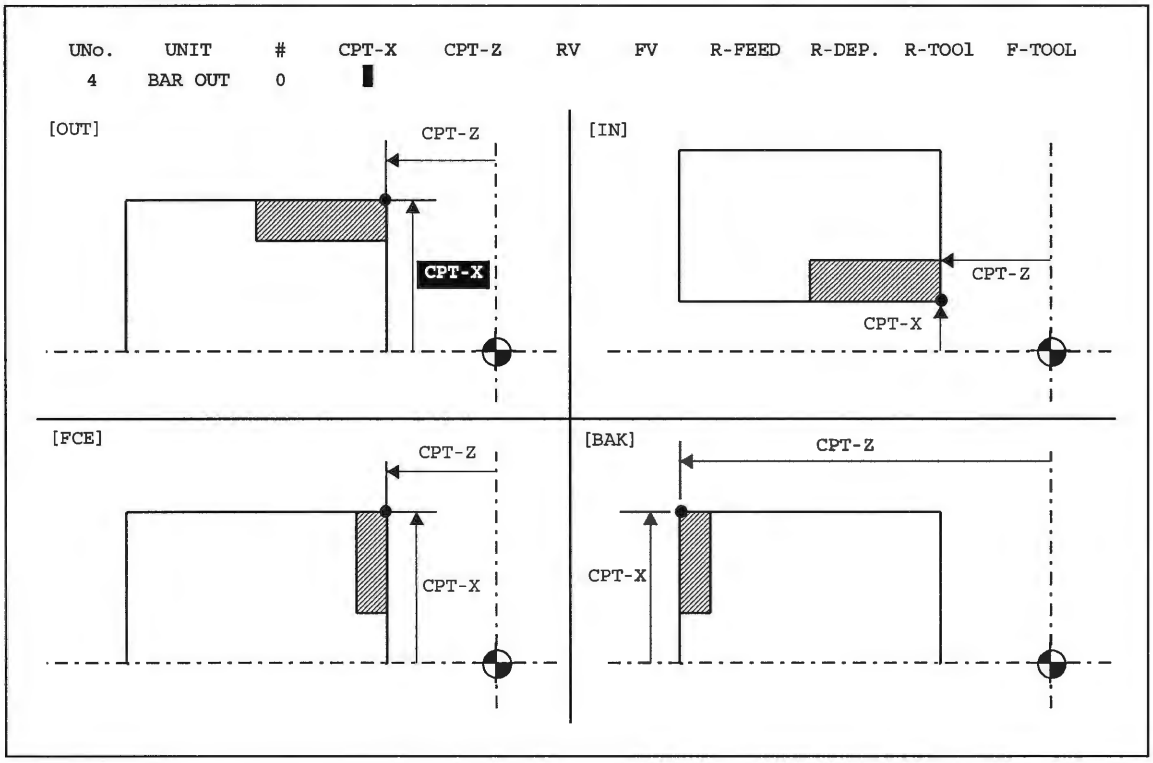

UNo. UNIT # CPT-X CPT-Z RV
1 BAR OUT 0 80. 0. 130
SEQ SHP S-CNR SPT-X SPT-Z FPT-X FPT-Z
1
Cursor
2 LIN □Ÿ □Ÿ 60. 45.
One blank line (sequence data line) will be inserted here.
UNo. UNIT # No. PITCH WIDTH FINISH RV
2 GRV OUT 0 1 0 5. 0. 120
SEQ S-CNR SPT-X SPT-Z FPT-X FPT-Z
1 60. 45. 56. 45.

6 PROGRAM CREATING/EDITING FUNCTIONS

6-1 Help Function

A help function is provided in the NC unit to give an illustrated description of program data. Help display shown below will be called up by pressing the **HELP** menu key with the cursor placed on an unit data item on the **PROGRAM** display. In the illustration the display of the respective item is reversed according to the cursor position.

Example: Help display for the bar-materials machining unit (BAR)



In the Help display, you can check details of the data to be set.

Note 1: Not all types of data can be plotted in the Help display. See the relevant section of this manual if you are placed at a loss what type of data to set in the program.

Note 2: Items which will be auto-set and those which will have an illustration on the menu display may not be indicated in the Help display.

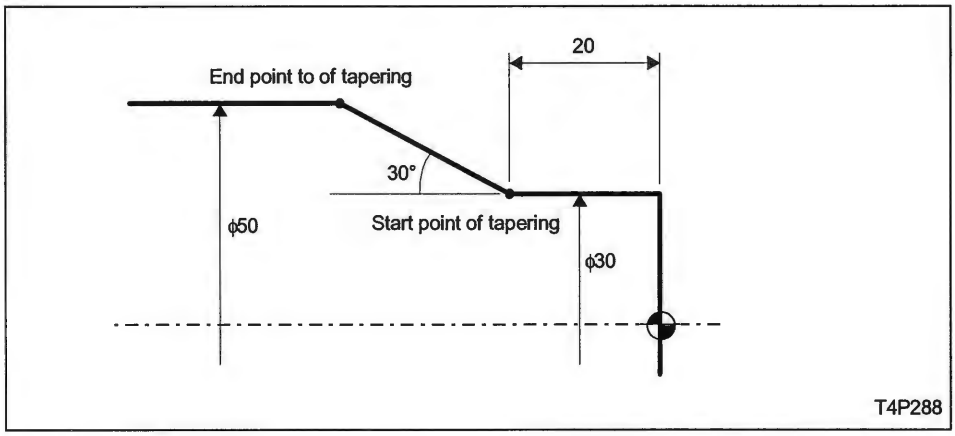
Note 3: The Help display cannot be called up unless the cursor is placed on an unit data line.

- Automatic calculation can also be performed in grafically checking the programmed data on the **TOOL PATH** or **SHAPE CHECK** display and the result is entered in a program.

Given below is the procedure of data setting for automatic calculation in cases 1 to 8 shown in the table above.

1. If start or end point of tapering is unknown.

Example: FPT-Z of tapering is unknown.

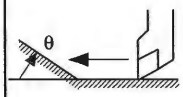
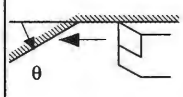
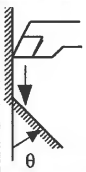
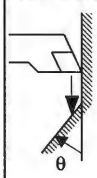
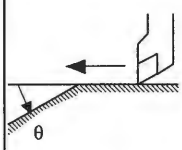
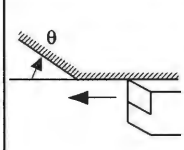
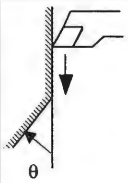
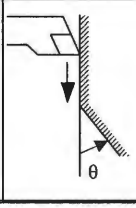


Set data as follows:

UNo.	UNIT	#	CPT-X	CPT-Z	RV	FV	R-FEED	R-DEP.
*	BAR OUT	*	* * *	* * *	* * *	* * *	* * *	* * *
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th
1	LIN		•	•	30.	20.		•
2	TPR		30.	20.	50.	?		30.

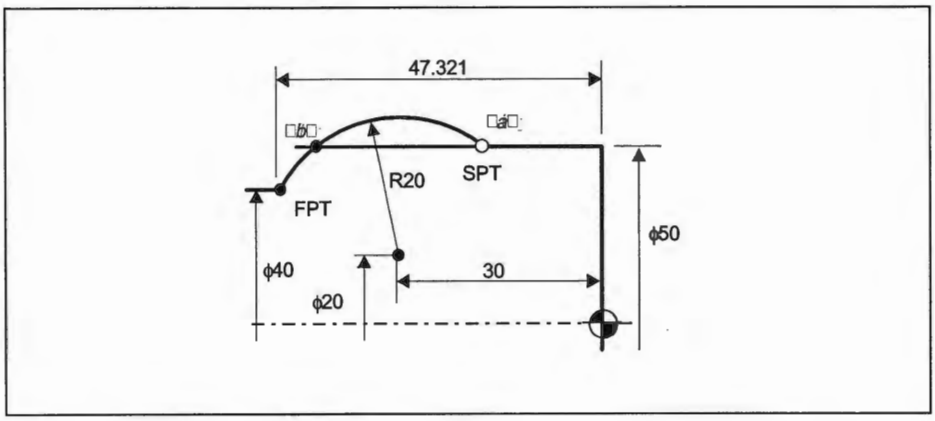
Press the **I. POINT?** menu key for the unknown FPT-Z.
Enter the tapering angle, 30°, for RADIUS/th.

Note: Enter positive angle value to designate upward tapering, or negative value for downward tapering.

	OUT•OUT•	IN•IN•	FCE•FCE•	BAK•BAK•
θ: Positive value				
θ: Negative value				

2. In general, an arc and a line cross each other at two points. To specify which one is to be set, use the menu keys **UP•**, **DOWN•**, **LEFT•** or **RIGHT•** on the CTR sequence line at the RADIUS/th item for unknown SPT or at ROUGH for FPT.

   Example:



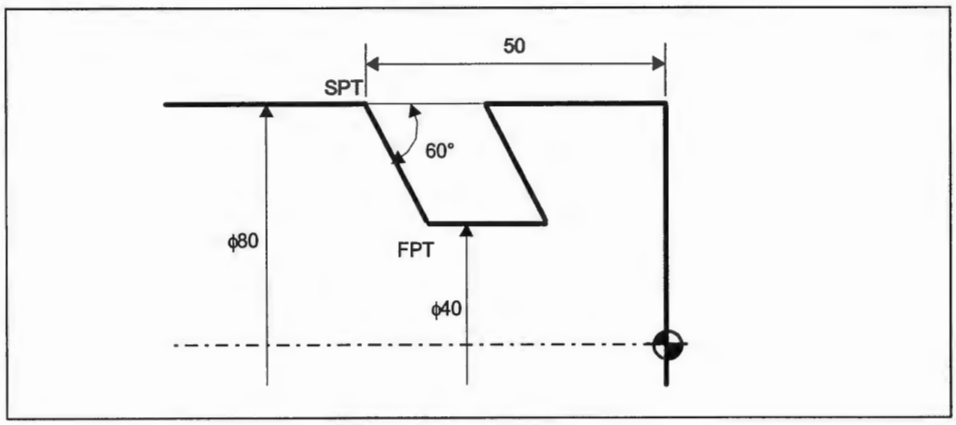
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	▟		50.	?	40.	47.321		20.	
2	CTR	•	20.	30.	•	•	•	•	

   To specify (*a*) for calculation of SPT-Z, press the **RIGHT•** menu key at RADIUS/th since the one point (*a*) lies on the right of the other possible point (*b*).

3. **If start or end point of tapered shape is unknown (for GRV unit).**

   As for the case 1, one of the items SPT-X to FPT-Z can be auto-set if the tapering angle is clearly known.

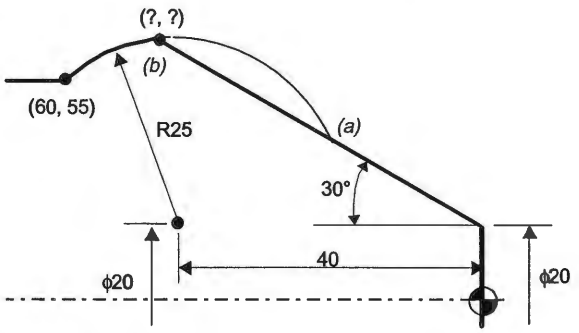
   Example: FPT-Z of tapering is unknown.



   Set data as follows:

UNo.	UNIT	#	No.	PITCH	WIDTH	FINISH	RV	FV
*	GRV OUT	0	1	0.	30.	•	•	120
SEQ	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANGLE	
1		80.	50.	40.	?		60.	

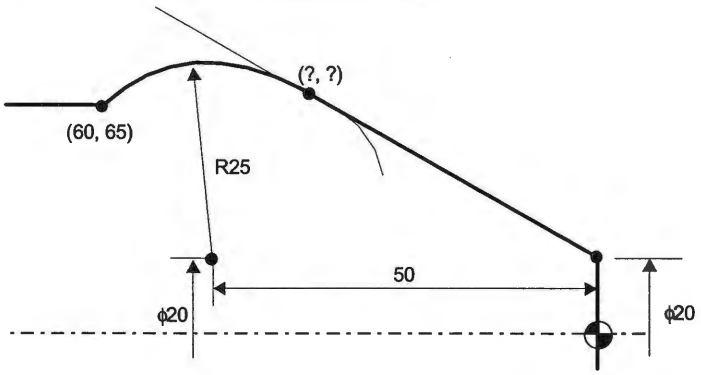
5. If intersecting point of tapering and arc is unknown.



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
TPR		20.	0.	?	? *1		30. *2	
⏚		?	? *1	60.	55.		25. *3	
CTR	□Ÿ	20.	40. *4	□Ÿ	□Ÿ	□Ÿ	□© *5	

*1. Press the **I. POINT?** menu key for unknown coordinates of the intersecting point of tapering and arc (⏚).
*2. Enter the tapering angle.
*3. Enter the radius of arc.
*4. Enter the coordinates of arc center.
*5. To specify (*b*) from among the two intersecting points of tapering and arc, press the **LEFT** □©(or **UP**□) menu key.

6. If osculation point of tapering and arc is unknown.



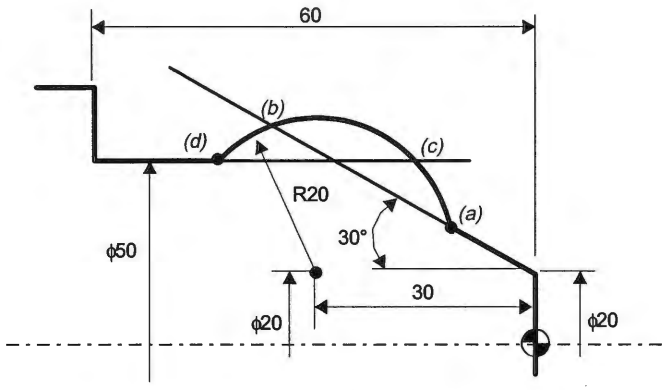
SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
TPR		20.	0.	? •	? • *1			
⏚		? •	? • *1	60.	65.		25. *2	
CTR	□Ÿ	20.	50. *3	□Ÿ	□Ÿ	□Ÿ		

*1. Press the **C. POINT** • **?** menu key for unknown coordinates of the osculation point of tapering and arc (⏚).
*2. Enter the radius of arc.
*3. Enter the coordinates of arc center.

9. Supplement

In cases 5 to 8, the following unknown items can also be auto-set.

Example: For intersecting point of tapering and arc, SPT-X or -Z of tapering and FPT-X or -Z of arc are unknown.



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
TPR		20.	0.	?	? *1		30. *2	
⊥		?	? *1	50.	? *3		20. *4	
CTR	□	20.	30. *5	□	□	□	□ *6	□ *7
LIN		□	□	50.	60.			

*1. Press the **I. POINT?** menu key for unknown coordinates of the intersecting point of tapering and convex arc.

*2. Enter the tapering angle.

*3. Press the **I. POINT?** menu key for unknown FPT-Z of the convex arc.
In general, even an unknown coordinate of arc end point can be calculated with the intersecting point of tapering and arc remaining unknown.

*4. Enter the radius of convex arc.

*5. Enter the center coordinates of convex arc.

*6, 7 Press the menu key **UP** • , **DOWN** , **LEFT** • or **RIGHT** • at the items RADIUS/th and ROUGH to specify one of the two possible intersecting points of arc and tapering.
Press at RADIUS/th the **RIGHT** • (or **DOWN** •) menu key to specify (*a*) from among the two intersecting points of tapering and convex arc.
Press at RGH the **LEFT** • menu key to specify (*d*) from among the two intersecting points of arc and straight line.

6-3 Automatic Cutting-Conditions Setting Function

Except for manual program machining units (MNP and MMP), the items of cutting conditions temporarily filled with "?" by pressing the **AUTO SET** menu key can automatically set upon specifying a tool for the respective unit. The automatic setting is performed using the data registered on the **CUTTING CONDITION** displays and other various parameters (refer to the table shown below for details of the calculation expressions).

If the programmed feedrate or surface velocity is modified using the VFC function, the new modified value will be stored together with the corresponding basic conditions (machining mode, materials type of workpiece and tool, outside diameter and length of workpiece) into the system memory. Those modified values will then be given priority in the next and subsequent auto-settings (and displayed in reverse form) if the basic conditions agree with the stored ones.

Calculation expressions of auto-setting function

Process	Calculation expressions
BAR CPY CNR EDG	RV = R-SPD in CUTTING CONDITION (TURNING) display × [R-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] FV = F-SPD in CUTTING CONDITION (TURNING) display × [F-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [F-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] R-FEED = R-FEED in CUTTING CONDITION (TURNING) display × [R-FEED%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-FEED%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] R-DEP = R-DEPTH in CUTTING CONDITION (TURNING) display × [R-DEP%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100 × [R-DEP%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100]
GRV	RV = R-SPD in CUTTING CONDITION (TURNING) display × [R-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] FV = F-SPD in CUTTING CONDITION (TURNING) display × [F-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [F-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] FEED = R-FEED in CUTTING CONDITION (TURNING) display × [R-FEED%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-FEED%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] DEP = R-DEPTH in CUTTING CONDITION (TURNING) display × [R-DEP%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-DEP%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100]
THR	HGT = Thread pitch × **K24**/10000 (when OUT/FCE/BAK is selected for metric threads) HGT = Thread pitch × **K25**/10000 (when IN is selected for metric threads) HGT = Thread pitch × **K26**/10000 (when OUT/FCE/BAK is selected for inch threads) HGT = Thread pitch × **K27**/10000 (when IN is selected for inch threads) NUMBER = (see the description given afterwards) V = F-SPD in CUTTING CONDITION (TURNING) display × [F-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [F-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] DEPTH = (see the description given afterwards)
DRL	DEP-1 = Hole diameter (DRL-DIA) × **K17**/100 DEP-2 = **U44**/A A = 1000 for metric system DEP-3 = **U46**/A 10000 for inch system V = R-SPD in CUTTING CONDITION (TURNING) display x [R-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100] FEED = R-FEED in CUTTING CONDITION (TURNING) display × [R-FEED%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [R-FEED%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100]
TAP	PITCH = Data based on JIS (Japan Industrial Standards); depends on the nominal diameter of the thread. V = F-SPD in CUTTING CONDITION (TURNING) display × [F-SPD%(WORK) in CUTTING CONDITION (PERCENTAGE) display/100] × [F-SPD%(TOOL) in CUTTING CONDITION (PERCENTAGE) display/100]

Details on calculation expressions for THR unit

- NUMBER

#0, **#0** Metric $\frac{3.4 \times LEAD}{MULTI} + 3.59$

Example of program

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	RPM	FIN-X	FIN-Z	WORK FACE		
0	S45C	100.	50.	100.	2000	0.2	0.1	0		
UNo.	UNIT	#	CPT-X	CPT-Z	RV	FV	R-FEED	R-DEP.	R-TOOL	F-TOOL
1	BAR OUT	0	100.	0.	130	160	0.35	3.375	•	•
UNo.	UNIT	#	CPT-X	CPT-Z	RV	FV	R-FEED	R-DEP.	R-TOOL	F-TOOL
2	BAR IN	0	50.	0.	110	130	0.3	3.0	•	
UNo.	UNIT	#	CPT-X	CPT-Z	RV	V	R-FEED	R-DEP.	R-TOOL	F-TOOL
3	BAR OUT	0	100.	70.	130	160	0.35	3.375	•	

1. **To auto-set a forward-cutting tool for the roughing of UNo. 1:**

   (1) Move the cursor to •.

   (2) Press the **AUTO SET** menu key.

   ➔ Tool No. 1 A will be auto-set at position •.

2. **To auto-set a forward-cutting tool for the finishing of UNo. 1:**

   (1) Move the cursor to •.

   (2) Press the **AUTO SET** menu key.

   ➔ Tool No. 3 will be auto-set at position •.

3. **To auto-set a forward-cutting tool for the roughing of UNo. 2:**
   (a case where tool auto-setting is impossible)

   (1) Move the cursor to •.

   (2) Press the **AUTO SET** menu key.

   - Auto-setting can not be executed since no BAR IN tools have been registered on the **TOOL DATA** display.

   ➔ Alarm **TOOL AUTO IMPOSSIBLE** will be displayed.

4. **To auto-set a reverse-cutting tool for the roughing of UNo. 3:**
   (change of left/right-handed tool identification)

   (1) Move the cursor to •.

   (2) Press the **REVERSE CUT** menu key.

   ➔ The REVERSE CUT menu item will be reversed.

   (3) Press the **AUTO SET** menu key.

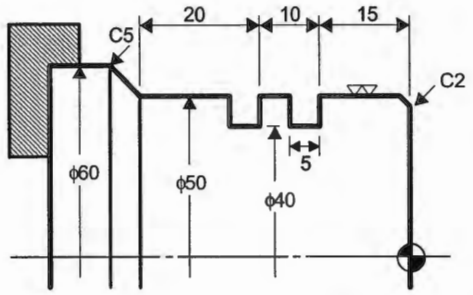
   ➔ Tool No. 5 will be auto-set at position •.

Note 1: Forward cutting refers to a direction towards the chuck (or the center line).
Reverse cutting refers to a direction away from the chuck (or the center line).

7 SAMPLE PROGRAMS

Example 1: 2-axes machining (BAR, GRV)

Machining drawing



Workpiece materials type: S45C
Workpiece size: ϕ60□60

T4P314

Program

```
UNo.  MAT       OD-MAX  ID-MIN  LENGTH  RPM    FIN-X   FIN-Z   WORK
0     S45C      60.     0.      60.     2000   0.2     0.1     FACE
                                                               0.
UNo.  UNIT      #         DEP-X   DEP-Z   RV    FV    R-FEED   R-DEP.   R-TOOL    F-TOOL
1     BAR OUT   0         60.     0.      130   200   0.3      2.5      1         2
SEQ   SHP       S-CNR     SPT-X    SPT-Z    FPT-X    FPT-Z    CNR/$    RADIUS/th RGH
1     LIN       C 2.      •        •        50.      50.      C 5.     •         ••4
UNo.  UNIT        #   No. PITCH   WIDTH   FINISH RV    FV    FEED    DEP.  R-TOOL   F-TOOL
2     GRV OUT     0   2   10.     5.      •      •     120   0.08    2.    •        3
SEQ           S-CNR       SPT-X    SPT-Z    FPT-X    FPT-Z    CNR      ANG       RGH
1                         50.      20.      40.      20.
UNo.  UNIT        COUNTER   RETURN    WK.No.   CONT.   NUM.   SHIFT
3     END         0         0                  0       0      0.
```

Note: The displayed workpiece material menu differs according to the data set at WORK MAT item in the **CUTTING CONDITION (REGIST)** display.

8 THREE-DIGIT G-FORMAT

8-1 Outline

The three-digit G-format is a format of expressing MAZATROL program data and other NC data. The various types of data within the NC unit are each assigned to a specific "three-digit G + address + data" set. Use of the data input/output functions based on the three-digit G-format allows the NC-stored data to be managed under the same environment as those of EIA/ISO programs. Data that have been output to external units in the three-digit G-format can be edited using a personal computer, and if the edited data are restored into the NC unit, the corresponding original data within the NC unit will be auto-modified according to the required edition.

8-2 Detailed Description

1. MAZATROL program data
   - Unit data have an assigned specific three-digit G-code for each unit.
   - Tool sequence data succeed the block of the three-digit G-code of the unit data and are positioned between code G426, which denotes the beginning of the sequence data, and code G427, which denotes the end of the sequence data.
   - Shape sequence data succeed the block of the three-digit G-code of the unit data and are positioned between code G420, which denotes the beginning of the sequence data, and code G421, which denotes the end of the sequence data.
   - The TPC data, if present, succeed the block of the three-digit G-code of the unit data and are positioned between code G422, which denotes the beginning of TPC data, and code G423, which denotes the end of TPC data.
   - Machining set-up information is keyed to the address that immediately succeeds code G424.
   - Process layout information is keyed to the address that immediately succeeds code G425.
   - Machining set-up data, machining program data and process layout data are output in that order.
     The order of output of the machining program data is also predetermined. That is, unit data, TPC data and tool sequence data and shape sequence data are output in that order. Do not change the order.
2. Data on the **TOOL OFFSET**, **TOOL DATA**, **TOOL LIFE**, **TOOL FILE**, **PARAMETER**, **VARIABLE**, **CUTTING CONDITION**, **CHUCK JAW DATA** and **WORK OFFSET** displays
   - The code G10 is used to input/output the above data.

The codes for data identification are listed up and described in detail on the following pages.

Note: The applicability of individual I/O-related functions (G-codes) depends upon the machine's specifications.

related address										Unit
FIN-X U	FIN-Z W	WORK FACE C	MPX D	FIN-LENGTH H						
										MTR
FV SF	R-FEED F	R-DEP. U	R-TOOL TR	F-TOOL TF						BAR
RV SR	FV SF	R-FEED F	R-DEP. U	R-TOOL TR	F-TOOL TF					CPY
R-DEP. U	R-TOOL TR	F-TOOL TF								CNR
R-TOOL TR	F-TOOL TF									EDG
MULTI R	HGT H	NUMBER K	C-SP S	DEPTH U	TOOL T					THR
FINISH I	RV SR	FV SF	R-FEED F	DEP. U	R-TOOL TR	F-TOOL TF				GRV
DEP-3 K	C-SP S	FEED F	TOOL T							DRL
PITCH F	C-SP S	TOOL T								TAP
										MNP
#6 MF	#7 MG	#8 (*2) MH/BH	#9 MI	#10 MJ	#11 MK	#12 (*2) ML/BL				M
DEP-2 J	DEP-3 K	C-SP S	FEED F	TOOL T						MDR
DEPTH H	DWELL X	PITCH F	C-SP S	TOOL T	# I	OD-MAX J	CHMF K	FR U	DEPTH R	MTP
FEED F	TOOL T									BOR
RV SR	FV SF	R-FEED1 U	R-FEED2 F	R-TOOL TR	F-TOOL TF					MGV
										MMP
FV SF	R-FR1 U	R-FR2 F	R-TOOL TR	F-TOOL TF						LCT
RV SR	FV SF	R-FR1 U	R-FR2 F	R-TOOL TR	F-TOOL TF					RGT
RV SR	FV SF	R-FR1 U	R-FR2 F	R-TOOL TR	F-TOOL TF					LFT
										SEP
SPDL J										TRS
										MES
SHIFT I										END
										SUB
C-SP S	FEED1/PITCH 1 F	FEED2/PITCH 2 E	TOOL T	TORNA. R	FR U	DIR. J				CIR

(*1) Materials name is expressed in parentheses.
Ex.: S45C → (S45C)

(*2) BD, BH and BL are used for the secondary miscellaneous function.
M-codes (m) for the opposite side are distinguished by an addition of 10000 to the original code number.

related addres								(Unit)
FPT-Z &Z	RADIUS R							MTR
SPT-Z Z	FPT-X &X	Crs. pnt. (Fpt) &D	FPT-Z &Z	F-CNR/$ &R/&C/K	RADIUS/θ R/E	RGH F(,F)		BAR
FPT-X &X	FPT-Z &Z	F-CNR/$ &R/&C/K	RADIUS/θ R/E	RGH F(,F)				CPY
F-CNR/$ &R/&C/K	RGH F(,F)							CNR
RGH F(,F)								EDG
								THR
FPT-Z &Z	CNR &R/&C	ANGLE D	RGH F(,F)					GRV
								DRL
								TAP
RAD./VARI. R/V	rpm/surf. vel. K/S	FEED F(,F)	M M	OFS H				MNP
SPT-Y/C V/C	CTR1/PITCH &Z/&P	CTR2 &Y	NUM. K	ANGLE D	TYPE E			MDR
SPT-Y/C V/C	CTR1/PITCH &Z/&P	CTR2 &Y	NUM. K	ANGLE D	TYPE E			MTP
SPT-Y/C V/C	CTR1/PITCH &Z/&P	CTR2 &Y	NUM. K	ANGLE D	TYPE E			BOR
FPT-R/X &I/&X	FPT-Z &Z	RGH F(,F)						MGV
RADIUS R	rpm/surf. vel. K/S	FEED F(,F)	M M	OFS H				MMP
FPT-Y/C &V/&C	RADIUS R	RGH F(,F)						LCT
FPT-Y/C &V/&C	RADIUS R	RGH F(,F)						RGT
FPT-Y/C &V/&C	RADIUS R	RGH F(,F)						LFT
BASE W								MES DIA
T LIM + U	T LIM – V	BASE W						MES STP
T LIM + U	T LIM – V	BASE W						MES GRV
T LIM + U	T LIM – V	BASE W						MES WID
T LIM + U	T LIM – V	BASE W						MES DIS
								MES TOL
RETURN V	MOVE W							MES ZOF
SHIFT-C U	DIR. L	RETURN V	MOVE W					MES COF
								MES EXT
ARGM 5 (*3)	ARGM 6 (*3)							SUB
SPT-Y/C V/C	CTR1/PITCH &Z/P	CTR2 &Y	NUM. K	ANGLE D	TYPE E			CIR

(Suppl.) The data of address A (SHP) are as follows:

MTR and Turning LIN : 1 ▟ : 4
TPR : 2 CTR : 5
▟ : 3

Milling PNT : 17 CW : 21
CRC : 18 CCW : 22
STP : 19 LIN : 23
LNE : 20

H	I	J	K	L	M	N	O	P	Q	R
Rm-code 1	Rm-code 2	Rm-code 3	Gear No. for roughing	Gear No. for finishing	Offset No. for roughing	Offset No. for finishing	X-coord. of fix. pnt. for roughing	X-coord. of fix. pnt. for finishing	Z-coord. of fix. pnt. for roughing	Z-coord. of fix. pnt. for finishing
Workpiece measure-ment flag	Tool-tip measure-ment flag	Distance for retrial								
Spindle speed	Pressing distance									
Decrease in feedrate, 45 wall	Decrease in feedrate, 90 wall									
Edge relief distance										
Last threading amount	Min. cutting amount	Chamfer. amount								
Returning speed	Returning destance	Overlap amount								
Ending feedrate for drilling										
Ending feedrate for drilling										
Relay point P1X	Relay point P1Z	Relay point P2X	Relay point P2Z	Relay point P3X	Relay point P3Z	TPC data No.				
Radial clearance										

F. Process layout data

G425 P_ N_ I_ M_ A_ B_ C_ K_ T_ D_;

P : Unit No.

I : MAZATROL ID

ID	Unit	ID	Unit	ID	Unit
1	Common	18	Corner-machining	33	Mill-tapping unit
2	Materials shape	19	Edge-machining	34	Boring unit
3	M-code	20	Threading	35	Mill-grooving unit
4	Measurement	21	Grooving	36	Mill-line center-machining unit
5	End	22	Drilling (turning mode)	37	Mill-line right-machining unit
11	Subprogram	23	Tapping (turning mode)	38	Mill-line left-machining unit
16	Bar-materials machining	24	Manual program machining	39	Mill-grooving unit
17	Copy-machining	32	Mill-drilling unit	40	Circular milling unit

M : TPS mode cancel YES (0) / NO (1)

A : Machining part Roughing (0) / Finishing (1)

B : Approach relay point Auto (0) / Manual (1)

C : Escape relay point Auto (0) / Manual (1)

D : Nominal diameter

K : Machining part code

N : Sequence No.

T : Tool

T (Tool shape code A). (Tool shape code B). (Tool direction). (Suffix)

Beginning of sequence data	G420
End of sequence data	G421
Beginning of TPC data	G422
End TPC data	G423
Machining set-up data	G424
Process layout data	G425

5. Data on the **TOOL FILE** display

   For end mill, face mill and ball end mill

   G10 **L39** C_D_S_(_____)R_H_A_N_P_;

   For chamfering cutter

   G10 **L39** C_D_S_E_(_____)A_N_P_;

C:	Tool name	D:	Nominal diameter	S:	Suffix
( ):	Material	R:	Infeed amount	H:	Number of blades
A:	Angle	E:	Minimum diameter	N:	No.
P:	Validity				

6. Data on the **TOOL LIFE** display

   G10 **L44** P_N_T_F_A_B_C_D_;

   * **L64** for tool data of HEAD 2

P:	Group No.	T:	Tool No.	F:	Offset No.
N:	Priority No.	A:	LIFE TIME	B:	LIFE NUM.
C:	USED TIME	D:	USED NUM.		

7. Data on the **CUTTING CONDITION (TURNING)** display (for turning units)

   G10 **L51** P_A_B_C_D_;

P:	No.	A:	R peripheral speed	B:	F peripheral speed
C:	R feedrate	D:	R infeed		

8. Data on the **CUTTING CONDITION (REGIST)** display (for workpiece material)

   G10 **L53** P_(_)A_B_C_D_;

P:	No.	A:	R peripheral speed	B:	F peripheral speed
C:	R feedrate	D:	R infeed	( ):	Material

9. Data on the **CUTTING CONDITION (REGIST)** display (for tool material)

   G10 **L54** P_(_)A_B_C_D_;

P:	No.	A:	R peripheral speed	B:	F peripheral speed
C:	R feedrate	D:	R infeed	( ):	Material

10. Data on the **CUTTING CONDITION** display (for point, linear, face units)

    G10 **L58** N_P_(____);

    G10 **L58** N_P_S_F_;

N:	No. for each tool	1 to 12
P:	No.	1 to 136
S:	Peripheral speed	0 to 65535
F:	Feedrate	0 to 99999 (mm/1000, inch/10000)
( ):	Tool material	8 characters

11. Data on the **CUTTING CONDITION** display (A, B, C parameter)

    G10 **L59** A_Z_;

    G10 **L59** B_Z_;

    G10 **L59** C_Z_;

A,B,C:	Address	1 to 108
Z:	Data	1 to 65535 (–32764 to 32767)

- NOTE -